

Using the Tormach PCNC 1100

Programmer's and Operator's Guide to the Personal CNC 1100



© 2005/6/7 Tormach LLC and John Prentice, Castlewood Consultants

Questions or comments?
Please email us at:
tech@tormach.com



PCNC 1100 Manual
Part Number 30625 – Rev C3-1

Contents

1.	Preface	1-1
1.1	Safety.....	1-1
1.1.1	Electrical Safety.....	1-1
1.1.2	General Operating Safety	1-1
1.1.3	Safety Publications	1-2
1.1.4	Safety Precautions.....	1-2
1.2	Personal CNC Concept	1-3
1.3	Performance Expectations	1-3
1.3.1	Cutting Ability.....	1-3
1.3.2	Understanding Accuracy	1-4
1.3.3	Resolution, Accuracy and Repeatability of the PCNC.....	1-4
1.4	Scope and Intellectual Property.....	1-4
1.5	Nomenclature.....	1-5
2.	Preparation	2-1
2.1	Planning for Your PCNC.....	2-1
2.1.1	Electrical Connection.....	2-1
2.1.2	Location and Mounting	2-1
2.1.3	Computer Mounting Arrangement.....	2-3
2.1.4	Learning and Training.....	2-3
2.2	Receiving, Unpacking and Checking Shipment.....	2-4
2.2.1	Moving the Crate	2-4
2.2.2	Uncrating and Inspection.....	2-4
2.3	Assembling Y-axis Drive.....	2-4
2.4	Mounting the PCNC	2-5
2.4.1	Lifting onto Stand	2-5
2.4.1.1	Lifting from Below	2-5
2.4.1.2	Lifting from Above	2-5
2.4.2	Fixing to Stand	2-6
2.4.3	Accessories.....	2-6
2.5	Power to the PCNC.....	2-6
2.6	Power for Machine Accessories	2-7
2.7	Control Computer and Software Installation.....	2-7
2.7.1	Computer Specification.....	2-7
2.7.2	Optimizing the Windows Installation.....	2-8
2.7.3	Installing the Control Software.....	2-9
2.7.3.1	Installing.....	2-9
2.7.3.2	Vital Re-boot	2-9
2.7.3.3	Testing the Installation	2-10
2.7.3.4	DriverTest After a Software Crash.....	2-11
2.7.3.5	Manual Driver Installation and Un-installation.....	2-11
2.8	Running the PCNC	2-11
2.8.1	Main Switch and Control Panel	2-12
2.8.2	Changing the Spindle Speed Range	2-14
2.8.3	Computer Control of the Spindle and Coolant.....	2-14
2.8.4	MDI for Entering G- and M-code Commands.....	2-15

2.8.5	Jogging the Axes.....	2-15
2.9	Summary	2-16
3.	Making Your First Part	3-1
3.1	Coordinates	3-1
3.1.1	Referencing the Machine.....	3-1
3.2	Loading a Demonstration Program.....	3-2
3.3	Running the Demonstration Part-program.....	3-3
3.3.1	Part Material	3-3
3.3.2	Setting Work Offsets.....	3-3
3.3.3	Cutting in Air.....	3-4
3.3.4	Cutting the Actual Part.....	3-4
3.3.5	Summary	3-5
4.	Routes from an Idea to a Part	4-1
4.1	Controlling the PCNC.....	4-1
4.2	Choosing the Appropriate Design Software	4-2
4.3	Software for CAD/CAM.....	4-2
4.3.1	3D CAD	4-3
4.3.2	2D CAD	4-3
4.3.3	CAM	4-4
4.3.4	Running the G-code	4-6
4.3.5	CAD/CAM Systems.....	4-6
4.4	Programming with Wizards	4-9
4.4.1	Machining Wizard Concept.....	4-9
4.4.1.1	Selecting and Running a Wizard.....	4-9
4.4.1.2	Standard Wizard Features.....	4-9
4.4.1.3	G-code from a Wizard.....	4-10
4.4.1.4	Commercial Wizards.....	4-10
5.	Machine Controls	5-1
5.1	Machine Operation	5-1
5.1.1	Operator's Panel	5-1
5.1.2	Tool Changing	5-3
5.1.2.1	Tooling Strategy	5-3
5.1.2.2	Changing R8 Tools	5-3
5.1.2.3	Changing TTS Tools	5-4
5.1.3	Spindle Speed Ranges.....	5-5
5.2	Screen Control Panels.....	5-5
5.2.1	Using the Screens	5-5
5.2.2	Families of Related Controls	5-5
5.2.2.1	Screen Switching Controls	5-5
5.2.2.2	Axis Control Family.....	5-6
5.2.2.3	Jogging Control Family.....	5-7
5.2.2.4	Spindle Speed Control Family	5-8
5.2.2.5	Feed Control Family	5-9
5.2.2.6	Program Running Control Family.....	5-10
5.2.2.7	Toolpath Control Family.....	5-11
5.2.2.8	File Control Family.....	5-12
5.2.2.9	Work Offset and Tool Table Control Family.....	5-13
5.2.2.10	MDI and Teach Control Family.....	5-13
5.2.2.11	Loop Control Family.....	5-13
5.2.2.12	Modes and Mode Alarm Control Family.....	5-14

Contents

5.2.2.13	Rotational Diameter Control Family	5-14
5.2.2.14	Toolchange Position Control Family.....	5-14
5.2.2.15	Inhibits and Overrides Control Family	5-15
5.2.2.16	Feeds and Speeds Calculator	5-15
5.2.2.17	Tapping Configuration Family.....	5-15
5.2.2.18	Misc. Settings Control Family	5-16
5.3	USB Jogging Pendants	5-16
5.3.1	Jog/Shuttle Controller	5-16
5.3.2	Keypad Pendant	5-17
6.	Using Multiple Tools	6-1
6.1	Offsets and Coordinate Systems	6-1
6.2	Measuring Techniques	6-2
6.2.1	“Roll-Your-Own” Gauge Method.....	6-2
6.2.2	Roller Gauge Method.....	6-3
6.2.3	Adjustable Parallel Method	6-4
6.2.4	Comments on Accuracy	6-4
6.3	Setting Work Offsets.....	6-5
6.3.1	Direct Entry to Axis DRO	6-5
6.3.2	Using the Touch Buttons.....	6-5
6.3.3	Setting X and Y Offsets	6-5
6.3.3.1	With a Probe	6-6
6.3.3.2	Measuring Off an Edge	6-6
6.3.3.3	Laser Centering Techniques	6-6
6.4	Tool Offsets	6-7
6.4.1	Tormach Tooling System	6-7
6.4.2	How Tool Offsets Work	6-7
6.4.3	Master Tool and Numbering Tools	6-9
6.4.4	Using the Probe and Tool Setter	6-10
6.4.5	Tool Length Offsets with TTS.....	6-10
6.4.6	Tool Length Offsets with General Tooling.....	6-11
6.4.7	Comments on Tool Offsets.....	6-11
6.5	Multiple Work Origins	6-12
6.5.1	G54 Work Offset	6-12
6.5.2	Other Work Offsets	6-12
6.6	Cutter Diameter Compensation.....	6-12
6.6.1	CAD/CAM and Wizards	6-12
6.6.2	Control Software Compensation.....	6-12
6.6.3	Two Kinds of Contour	6-13
6.6.3.1	Material Edge Contour	6-13
6.6.3.2	Toolpath Contour	6-14
7.	Part-programming Language Reference.....	7-1
7.1	Definitions	7-1
7.1.1	Control Software.....	7-1
7.1.2	Linear Axes	7-1
7.1.3	Rotational Axes	7-1
7.1.4	Scaling Input.....	7-1
7.1.5	Controlled Point.....	7-1
7.1.6	Coordinated Linear Motion	7-2
7.1.7	Feed Rate.....	7-2
7.1.8	Arc Motion	7-2
7.1.9	Coolant.....	7-3
7.1.10	Dwell.....	7-3
7.1.11	Units.....	7-3

7.1.12	Current Position	7-3
7.1.13	Selected Plane.....	7-3
7.1.14	Tool Table	7-3
7.1.15	Path Control Modes	7-3
7.2	Interpreter Interaction with Controls	7-4
7.2.1	Feed and Speed Override controls	7-4
7.2.2	Block Delete Control	7-4
7.2.3	Optional Program Stop Control	7-4
7.3	Tool File	7-4
7.4	Part-programs Language.....	7-4
7.4.1	Overview.....	7-4
7.4.2	Parameters.....	7-5
7.4.3	Coordinate Systems	7-5
7.5	Formatting Code Lines (Block)	7-5
7.5.1	Line Number.....	7-5
7.5.2	Subroutine Labels	7-6
7.5.3	Word	7-7
7.5.3.1	Number.....	7-7
7.5.3.2	Parameter Value.....	7-7
7.5.3.3	Expressions and Binary Operations	7-7
7.5.3.4	Unary Operation Value	7-8
7.5.4	Parameter Setting.....	7-8
7.5.5	Comments and Messages	7-8
7.5.6	Item Repeats	7-9
7.5.7	Item Order	7-9
7.5.8	Commands and Machine Modes.....	7-9
7.6	Modal Groups	7-10
7.7	G-codes.....	7-11
7.7.1	Rapid Linear Motion – G00.....	7-11
7.7.2	Linear Motion at Feed Rate – G01.....	7-13
7.7.3	Arc at Feed Rate – G02 and G03	7-13
7.7.3.1	Radius Format Arc.....	7-13
7.7.3.2	Center Format Arc	7-14
7.7.4	Dwell – G04	7-15
7.7.5	Coordinate System Data Tool and Work Offset Tables – G10.....	7-15
7.7.6	Clockwise/Counterclockwise Circular Pocket – G12 and G13.....	7-16
7.7.7	Exit and Enter Polar Mode – G15 and G16.....	7-16
7.7.8	Plane Selection – G17, G18 and G19.....	7-16
7.7.9	Length Units – G20 and G21.....	7-17
7.7.10	Return to Home – G28 and G30	7-17
7.7.11	Reference Axes – G28.1.....	7-17
7.7.12	Straight Probe – G31.....	7-17
7.7.12.1	Straight Probe Command	7-17
7.7.12.2	Using the Straight Probe Command.....	7-18
7.7.12.3	Example Code.....	7-18
7.7.13	Cutter Radius Compensation – G40, G41 and G42	7-19
7.7.14	Tool Length Offsets – G43, G44 and G49	7-19
7.7.15	Scale Factors – G50 and G51	7-20
7.7.16	Temporary Coordinate System Offset – G52	7-20
7.7.17	Move in Absolute Coordinates – G53	7-20
7.7.18	Select Work Offset Coordinate System – G54 to G59 & G59 P~.....	7-21
7.7.19	Set Path Control Mode – G61 and G64.....	7-21
7.7.20	Coordinate system rotation – G68 and G69.....	7-21
7.7.21	Canned Cycle – High Speed Peck Drill – G73	7-21
7.7.22	Cancel Modal Motion – G80	7-22
7.7.23	Canned Cycles – G81 to G89	7-22
7.7.23.1	Preliminary and In-Between Motion	7-23

Contents

7.7.23.2	G81 Cycle.....	7-23
7.7.23.3	G82 Cycle.....	7-24
7.7.23.4	G83 Cycle.....	7-24
7.7.23.5	G85 Cycle.....	7-25
7.7.23.6	G86 Cycle.....	7-25
7.7.23.7	G88 Cycle.....	7-25
7.7.23.8	G89 Cycle.....	7-25
7.7.24	Distance Mode – G90 and G91.....	7-26
7.7.25	G92 Offsets – G92, G92.1, G92.2 and G92.3.....	7-26
7.7.26	Feed Rate Mode – G93, G94 and G95.....	7-27
7.7.27	Canned Cycle Return Level – G98 and G99.....	7-27
7.8	Built-in M-codes.....	7-27
7.8.1	Program Stopping and Ending – M0, M1, M2 and M30.....	7-27
7.8.2	Spindle Control – M3, M4 and M5.....	7-28
7.8.3	Tool change – M6.....	7-29
7.8.4	Coolant Control – M7, M8 and M9.....	7-29
7.8.5	Re-run from First Line – M47.....	7-29
7.8.6	Override Control – M48 and M49.....	7-29
7.8.7	Call Subroutine – M98.....	7-29
7.8.8	Return from Subroutine – M99.....	7-30
7.9	Application Defined M-codes.....	7-30
7.9.1	Self-reversing Tapping Cycles.....	7-30
7.9.2	Goto Toolchange Position – M998.....	7-30
7.9.3	User Defined M-codes.....	7-30
7.10	Other Input Codes.....	7-31
7.10.1	Feed Rate – F.....	7-31
7.10.2	Spindle Speed – S.....	7-31
7.10.3	Select Tool – T.....	7-31
7.11	Order of Execution.....	7-31
7.12	Error Handling.....	7-31
8.	Machine Upgrades and Configuration.....	8-1
8.1	Fourth Axis – Rotary Table.....	8-1
8.1.1	Installing the Electronics.....	8-1
8.1.2	Utilizing the Fourth Axis.....	8-1
8.1.2.1	Referencing and Zeroing the Fourth Axis.....	8-1
8.1.2.2	Diameter Compensation Feature.....	8-1
8.1.3	Fourth Axis Applications.....	8-1
8.1.3.1	Engraving on a Periphery of a Cylinder.....	8-2
8.1.3.2	Gear Cutting.....	8-2
8.2	Digitizer Probes.....	8-3
8.2.1	Introduction to Uses of Probes and Tool Setters.....	8-3
8.2.2	Probing for Work/Tool Setting.....	8-4
8.2.2.1	Simple X/Y Probing.....	8-4
8.2.2.2	Z Probing.....	8-6
8.2.2.3	Comprehensive X/Y Probing.....	8-8
8.2.2.4	Probe Calibration.....	8-11
8.2.3	Digitizing parts from a model or for reverse engineering.....	8-12
8.2.4	The Probe Electrical Interface.....	8-12
8.3	Auto-reverse tapping.....	8-13
9.	Warranty, Specifications and Maintenance.....	9-2
9.1	Intended Use Statement.....	9-2

Contents

9.2	Support.....	9-2
•	This manual – ALWAYS the first place to check!!	9-2
•	Related documents found at: http://www.tormach.com/documents.htm	9-2
•	Our website at: www.tormach.com	9-2
•	Email to: info@tormach.com	9-2
•	Telephone Tormach at: 608-849-8381	9-2
•	Fax Tormach at: 209-885-4534.....	9-2
9.3	Outside of the Scope of Intended Use	9-2
9.4	Specifications.....	9-3
9.4.1	Mechanical	9-3
9.4.2	Electrical	9-3
9.4.3	System	9-4
9.4.4	Options.....	9-4
9.5	Maintenance.....	9-4
9.5.1	Protecting from Rust	9-4
9.5.2	Lubrication	9-4
9.5.3	Way Covers	9-5
9.5.4	Z-axis Adjustment.....	9-5
9.5.5	Adjusting Ballscrew Thrust Bearing Preload.....	9-6
9.5.5.1	Understanding Preloaded Angular Contact Bearings	9-7
9.5.5.2	Making the Adjustment.....	9-8
9.5.6	Speed Calibration.....	9-9
9.5.7	Using a Non-standard Printer Port	9-11
9.5.8	Troubleshooting.....	9-13
9.5.8.1	Mechanical	9-13
9.5.8.2	Electrical	9-14
9.6	Preparation for Transport.....	9-14
9.7	Disassembly for Transport	9-15
10.	Appendices.....	15
10.1	Appendix 1 – Not Used.....	15
10.2	Appendix 2 – Exploded Parts Views.....	15
10.3	Appendix 3 – Optimization of Windows XP.....	25
10.3.1	Remove Unnecessary Services and Startup Programs	25
10.3.2	Disable Power Management.....	26
10.3.3	Disable Automatic Updates.....	26
10.3.4	Set Computer to Standard PC not ACPI PC	26
10.4	Revision history.....	27
11.	Index.....	28

1. Preface

1.1 Safety



Any machine tool is potentially dangerous. Computer controlled machines are potentially more dangerous than manual ones because, for example, a computer is quite prepared to plunge a 3" diameter facing cutter at 50 inches per minute into a block of high-carbon steel or to mill the clamps off your table.

The PCNC 1100 can deliver sufficient force to break brittle tools, to crush bones and to tear flesh.

This manual tries to give you guidance on safety precautions and techniques but because we do not know the details of your workshop or other local conditions we can accept no responsibility for the performance of the machine or any damage or injury caused by its use. It is your responsibility to ensure that you understand the implications of what you are doing and to comply with any legislation and codes of practice applicable to your country or state.

1.1.1 *Electrical Safety*



Dual Power Input: The PCNC 1100 has two electrical power inputs. The primary supply is 230 VAC and is used for all axis and spindle motion. The secondary supply is 115 VAC. The secondary supply is used to provide power to the accessory outlets only and is not used for machine control. Either power supply can provide lethal electrical shocks. Both power inputs should be unplugged before working in the electrical cabinet.

Grounding: Both primary and secondary power inputs must be grounded. During installation it is not enough to assume that the ground line of a wall outlet is properly grounded. Check continuity between the machine frame and true earth ground (water pipe or similar) to ensure a good ground connection.

A Ground Fault Interrupt or GFI (i.e., Residual Current Circuit Breaker or RCCB in Europe) outlet must be used to supply the power to the 115 VAC power input. Your computer, monitor and coolant system are not bolted to the machine frame so proper grounding cannot be assumed. The combination of electrical power and water based coolant systems makes the GFI protection very important.

Electrical Panel: NEVER operate the machine tool with the cabinet door open. NEVER allow a coolant pump to operate with the cabinet door open. DO NOT allow the coolant system to flow coolant directly at the cabinet door seal or on the operator console controls. Neither the cabinet door seal nor the electrical controls are sealed against liquids.

Retained Electrical Power: Electronic devices within the electrical cabinet may retain dangerous electrical voltages after the power has been removed.

Electrical Service: Certain service and troubleshooting operations require access to the electrical cabinet while the electrical power is on. Only qualified electrical technicians should perform such operations.

1.1.2 *General Operating Safety*



Safe operation of the machine depends on its proper use and the precautions taken by each operator.

Read and understand this manual. Be certain every operator understands the operation and safety requirements of this machine before operating the machine.

Always wear safety glasses and safety shoes.

Always stop the spindle and check to ensure the CNC control is in the stop mode before changing or adjusting the belt/pulley position, tool or work piece.

Never wear rings, watches, gloves, long sleeves, neckties, jewelry or other loose items when operating or working around the machine. Long hair should be bound or kept under a hat.

Use adequate safeguarding around the operating envelope. It is the responsibility of the employer to provide and ensure point of operation safeguarding per OSHA 1910.212 – Milling Machine.

1.1.3 Safety Publications

Tormach recommends the following publications for assistance in enhancing the safe use of this machine.

- Safety Requirements for The Construction, Care and Use of Drilling, Milling and Boring Machines (ANSI B11.8-1983). Available from The American National Standards Institute, 1430 Broadway, New York, New York 10018.
- Concepts and Techniques of Machine Safeguarding (OSHA Publication Number 3067). Available from The Publication Office – O.S.H.A., U.S. Department of Labor, 200 Constitution Avenue, NW, Washington, DC 20210.



1.1.4 Safety Precautions

1. Do not run this machine without knowing the function of every control key, button, knob or handle. Refer to the manual or contact Tormach if any function is not understood.
2. Protect your eyes. Wear approved safety glasses (with side shields) at all times. You should never use compressed air to remove chips or to clean the machine. An air blast will often launch a metal chip into a place it should not be.
3. Avoid moving parts. Before operating this machine remove all jewellery including watches and rings, neckties and any loose-fitting clothing.
4. Keep your hair away from moving parts.
5. Take off gloves before you operate the machine. Gloves are easily caught in moving parts or cutting tools.
6. Never operate with unbalanced tooling or spindle fixtures.
7. Remove all tools (wrenches, chuck keys, etc.) from the spindle and machine surface before you begin. Loose items can become dangerous flying projectiles.
8. Use adequate work clamping. Do not allow your work piece to become a projectile.
9. Never operate a milling machine after consuming alcoholic beverages or taking strong medication.
10. Protect your hands. Stop the machine spindle and ensure that the computer control is stopped before you:
 - Change tools;
 - Change parts or adjust the work piece;
 - Change the belt/pulley position;
 - Clear away chips, oil or coolant – always use a chip scraper or brush;
 - Make an adjustment to the part, fixture, coolant nozzle or take measurements;
 - Remove protective shields or safeguards – do not reach for the part, tool or fixture around a guard.
11. Keep work area well lit. Ask for additional light if needed.
12. Keep the computer area clear of clutter. Recognize that machine motion can occur when certain keys are pressed. Objects falling on the keyboard can result in unexpected motion.

13. Avoid getting pinched in places where the table, saddle or spindle head create “pinch points” while in motion.
14. Securely clamp the work piece in a vise, on the table or in the fixture. Use proper holding clamping attachments and position them clear of the toolpath. Be aware of larger pieces that will be cut free during operations – loose parts can become projectiles.
15. Always use proper feeds and speeds, as well as depth and width of cut, to prevent tool breakage.
16. Use proper cutting tools for the job.
17. Do not use dull or damaged cutting tools. They break easily and become dangerous projectiles. Never use longer or larger tools than necessary.
18. Chips and dust from certain materials (e.g., magnesium) can be flammable. Fine dust from normally non-flammable materials can be flammable or even explosive.
19. Chips and dust from certain materials can be toxic. Vapours from certain overheated materials can be toxic. Always check a Materials Safety Data Sheet (MSDS) of suspect materials. Refuse machining work requests of unknown materials.
20. If you are in any doubt you must seek guidance from a professionally qualified expert rather than risk injury to yourself or to others.

1.2 Personal CNC Concept

The PCNC 1100 is a machine tool intended to make CNC machining more personal. As with the evolution of personal computers, the evolution of personal CNC alters the paradigm of what a machine tool is about. We aim for a machine tool so affordable that anyone can have one.

We feel that the work of engineers, inventors, technicians, hobbyists, educators and others will be enhanced when they have access to CNC machinery. In education, each student can run his own machine instead of waiting in line when the machine tool costs less than 20% of a small machining center. In R & D, turn-around on prototype design takes minutes instead of days when a machine is “at the ready” and on site. In general engineering, designs sent to the production machine shop are improved when the design engineer has been more involved in the prototype creation.

The PCNC 1100 offers the precision of a production machine but with cost/performance optimized for short run operation.

1.3 Performance Expectations

1.3.1 Cutting Ability

The machine is capable of cutting most materials at or near their recommended feeds and speeds. For example, for fast metal removal on 6061 aluminium we will run a $\frac{1}{2}$ " diameter 2 flute cutter at around 18 IPM (inches per minute) and 3000 RPM, using a full $\frac{1}{2}$ " depth of cut.; that is a pretty good volumetric rate of metal removal so it is essential to clear chips with a flood coolant. We will run smaller cutters when we are not trying to remove large amounts in a hurry. For most aluminium work we use $\frac{3}{8}$ ". The example above, using a $\frac{1}{2}$ " cutter, results in a surface speed of 390 SFM (surface feed per minute), a $\frac{3}{8}$ " cutter needs 4000 RPM to get the same surface speed, well within the performance envelope of the machine.

Cutting steel and iron needs a lower volumetric rate, thus slower feed and speed. The PCNC 1100 will run best using smaller cutters when working with tougher materials. For example, the general machining recommendation for some oil hardening steels is 30 SFM. Doing this with a $\frac{1}{2}$ " end mill, the surface speed calculation indicates 230 RPM, but that is below the minimum spindle speed of the PCNC 1100. By switching to a $\frac{1}{4}$ " end mill the recommended spindle speed becomes 460 RPM, well within the capability of the PCNC 1100. By keeping close to general machining recommendations your tools will last longer and you will have a better cut.

1.3.2 **Understanding Accuracy**

While a machine tool may seem absolutely rigid, the truth of the matter is that everything has some elasticity. Related to elasticity is the compressibility of components such as ball nuts and bearings. Preloading of bearings and ballscrews can remove the physical open space between moving parts, but the technique cannot eliminate compressibility. The key to achieving maximum accuracy is understanding and controlling the magnitude and direction of forces. Maximum accuracy is achieved when the forces are minimized, as occurs in a finishing cut. Maximum repeatability is achieved when the forces are repeatable, both in magnitude and direction.

1.3.3 **Resolution, Accuracy and Repeatability of the PCNC**

The minimum discrete position move is 0.0001", this is the resolution of motion. Machine accuracy is closely related to ballscrew accuracy. Our ballscrews are accurate to 0.0006" per foot, but considering all the other factors that come into play, we prefer to keep accuracy expectations to 0.0013" per foot. Repeatability will be better than 0.001" per foot.

Machining is a mix of science, skill and art. The caveat in stating accuracy and repeatability is that these factors depend on the techniques used by the machinist. A skilled machinist can often deliver accuracy that exceeds the accuracy specified by the machine builder, while an inexperienced machinist may have difficulty delivering the expected accuracy. With this understanding, we cannot tell you what accuracy you will be able to achieve in your own work. Nevertheless, the accuracy specified by a machine builder remains an important reference point.

1.4 **Scope and Intellectual Property**

This document is intended to give enough details about how to install, setup and use your PCNC 1100. It assumes that you have appropriate experience and/or access to training for any Computer Aided Design/Manufacture software that you intend to use with the machine. This document assumes familiarity with using typical Microsoft Windows applications programs as the control software for the PCNC runs under the Windows operating system.

Tormach LLC is dedicated to continual improvement of its products, so suggestions for enhancements, corrections and clarifications will be gratefully received.

Tormach LLC, Art Fenerty and John Prentice assert their right to be identified as the authors of this work. The right to make copies of this manual is granted solely for the purpose of training courses related to, evaluation of and/or use of the PCNC 1100. It is not permitted, under this right, for third parties to charge for copies of this manual beyond the cost of printing.

Every effort has been made to make this manual as complete and as accurate as possible but no warranty or fitness is implied. The information provided is on an "as is" basis. The authors and publisher shall have neither liability nor responsibility to any person or entity with respect to any loss or damages arising from the information contained in this manual.

Windows 2000, Windows XP and Windows Vista are registered trademarks of Microsoft Corporation. If other trademarks are used in this manual but not acknowledged please notify Tormach LLC so this can be remedied in subsequent editions.

1.5 Nomenclature

This manual uses the following typographical nomenclature:

Software control

Refers to a Control Software “soft” control. (i.e., a Windows control on the PC screen).

Hardware Control

Refers to a physical button or switch on the Operator’s Panel of the machine.

G-code (e.g., G01X34.8)

Used to show G-code programs.

Key name (e.g., *Enter*)

Tells you to press the indicated key.

2. Preparation

This chapter describes the work required to unpack and to commission the hardware and software of the PCNC.

It contains a lot of detail but can be completed in one or two hours by a person familiar with CNC machines. Enough detail is given here so that a beginner should be successful but some users may prefer to arrange for a machine tool expert to do this work.

If your machine has already been set-up then you can skip this chapter

2.1 Planning for Your PCNC

2.1.1 *Electrical Connection*

The PCNC 1100 is shipped with a 3-wire cord and no electrical plug. There are several different NEMA (National Electric Manufacturers Association) and non-NEMA plug patterns that can be used. Straight blade patterns are common in household use; twist-lock patterns are more common in industrial locations. Power required is 220 to 230 VAC, 50 or 60 Hz. Continuous current is below 15 amps, but a 20 amp breaker or slow blow fuse is recommended.

Both primary and secondary power inputs must be grounded. During installation it is not enough to assume that the ground line of a wall outlet is properly grounded. Check continuity between the machine frame and true earth ground (water pipe or similar) to ensure a good ground connection.



2.1.2 *Location and Mounting*

People experienced with CNC machining will undoubtedly have ideas as to how they want to setup their PCNC 1100 (figure 2.1). While the machine can be configured in many different ways, there are a few limitations. Many fully enclosed vertical machining centers incorporate high volume coolant systems that make the inside of the machine look like the inside of a dishwasher. The PCNC electrical cabinet and operator console should not be exposed to such conditions. Additionally, there should never be an enclosure or accessory that limits access to the emergency stop. Please keep these limitations in mind when you plan your configuration.

If your prior experience is limited to manual mills then keep in mind that, as CNC dramatically extends your machining capabilities, it will also change the way you cut metal. When your metal cutting is done by turning handles on a manual mill your operations will generally be limited to cleaning up a surface, drilling a hole pattern or cutting to



Figure 2.1 – An example mounting on stand

Preparation

a dimensional outline. With manual milling many people are accustomed to dry cutting, clearing chips with a small brush as they go.

With CNC you have a whole new world open to you. In many cases you may turn the majority of the stock into chips, cutting a shape out of a solid block of metal the way Michelangelo would cut a sculpture from a block of marble. Unless you are limiting yourself to cutting cast iron, wood, printed circuit boards or certain other materials, you will probably want a coolant system on your machine. Mist coolant can be effective for keeping your cutting tools cool, but it does little for clearing chips. Flood coolant will cool the cutting tools while clearing chips, but is more challenging to contain. The table of the PCNC has drain slots and a hole tapped for a pipe fitting to allow coolant collection, as is common on most small mills. Nevertheless, CNC machining operations commonly produce so many chips that you simply cannot keep the table drain running. At times, you may need a coolant flow that is simply too much for the table drain. The little drain tray that is common under manual machines or the open frame setup of a Bridgeport style knee mill just does not make it. We strongly recommend that you plan your setup with a full motion tray, such that coolant will be captured as it overflows the machine table within the full operating envelope of the machine.

Another reason to use a full motion tray is to reserve the space that will be required when the machine moves. If you use a narrow drip tray or none at all, you should plan for full machine motion **plus** some human space when you place the machine in your workshop. You do not want to locate it where you can create crush points between the machine table and a wall. When in operation, the X, Y and Z motions will **not stop** when they hit something. The machine will move with hundreds of pounds of force, enough to punch through a wall, tip over the machine or crush someone in the way.

Machine safety is the responsibility of the operator. This includes all aspects of safety: setup, location, operation, security and all other factors that involve safety.

The PCNC 1100 requires a minimum plan area of 67" wide by 43" deep. This gives clearance for the full motion of the table and for minimal access for cabling etc. The overall height required is 82" assuming that it is installed with the table at a working height of 36".

Tormach offers a range of stands, both ready-made and designs for you to have constructed locally.

You should choose a well lit location and provide any additional task-lighting to make it easy to setup work on the table.

Over time you will find that you accumulate a range of tools and tool holders so you should allocate space for storage of these near the machine. A rack with numbered slots is convenient

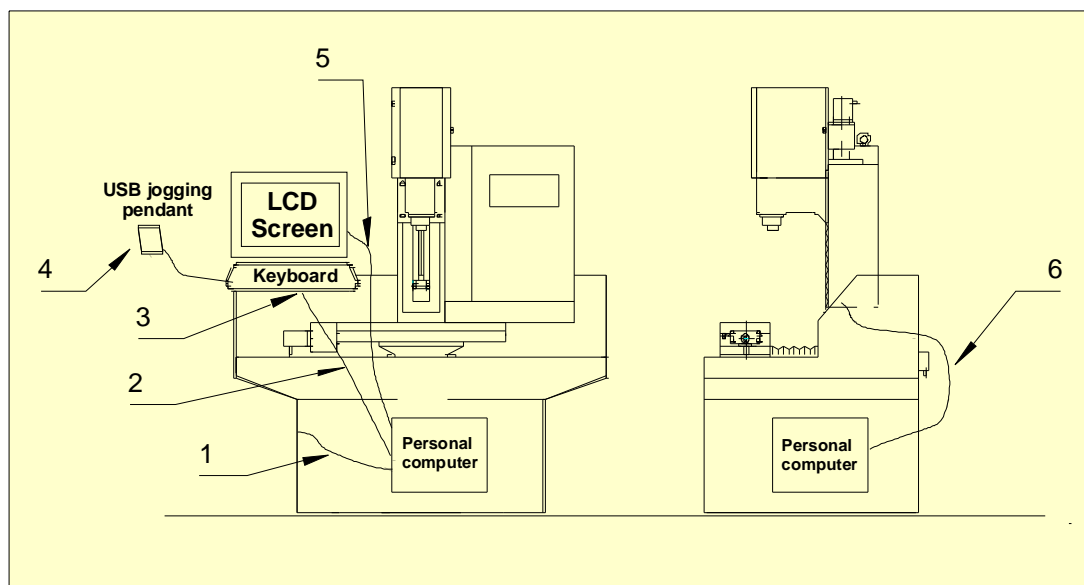


Figure 2.2 – Computer and display

to avoid errors when doing tool changes during a job.

2.1.3 Computer Mounting Arrangement

Keep the computer in a clean location, preferably inside the stand of the milling machine. Resist the temptation to expose the computer in any way. Providing access to floppy disks, CDs or direct computer controls will also open the computer to contamination and risk. Tormach offers accessories that will allow you to operate the system without exposing your computer.

While there are many possible configurations for your machine control computer, we suggest the following (figure 2.2):

1. USB bulkhead (panel mount) cable. This allows you to mount a USB socket directly on the side of the cabinet. You can use a standard USB flash drive to transfer G-code programs and other files to the machine controller. This is Tormach PN 30278 (USB bulkhead mount cable – 3' Version 2.0 USB A to A extension M-F).
2. USB extension cable, extending the short cable normally found on keyboards and other USB devices. Tormach PN 30279 (10FT USB 2.0 A to A Male/Female Extension Cable).
3. USB multimedia mini-keyboard with integral 2-port USB hub. This is about the size of most laptop keyboards. The keyboard includes a key which will power down the computer, allowing a convenient way to shutdown the system. It also has two USB ports. These can be used for a USB mouse, USB flash drive or one of the PCNC jogging pendants. The keyboard is not sealed against coolant or chips. This is Tormach PN 30284 (Mini Media USB Keyboard w/hub).
4. A Tormach USB jogging pendant is a very useful accessory for jogging, manual operations and machine setup. Two options are available: a key based pendant (Tormach PN 30214 Pendant, 10 key USB keypad) and a jog/shuttle controller pendant that gives very fine control of jogging speeds and distances. (Tormach PN 30616 Jog/Shuttle Controller).
5. LCD signal cables are normally too short. Most inexpensive VGA signal extension cables create serious signal degradation. This is particularly true with Super VGA screen resolutions. Tormach PN 30280 (10FT SVGA Super VGA M/F Monitor Cable w/ ferrites) is designed to extend Super VGA signals without degradation.
6. This is simply the AC power cord of the computer. The Computer switch on the operator console controls a convenience outlet on the bottom of the machine control cabinet. If you set the BIOS/CMOS configuration in your computer to start the computer when it sees AC power then the console switch will allow you to start the computer from the console. You should not shut off the computer from this switch due to issues with the Microsoft Windows operating system, but you can turn the computer on from the keyboard/screen controls.

The machine itself requires a 230 volt single phase wall power outlet rated at 6 amps, 13 amps inrush. You should also provide a separate wall outlet for 115 or 230 volts, depending on your locality, to act as power source for the computer, monitor and coolant pump (if used) which should be specified to suit you local voltage.

2.1.4 Learning and Training

The final element of planning your installation is to consider the training that you and any other users of the machine will need.

This manual will give you the basic information required to start manufacturing components with you PCNC. You must, however, expect to have to invest time in learning how to achieve the best results. The areas which you will find easy and those which will require more effort will of course depend on your background; you might be most comfortable with machining or with component design or even with information technology.

We believe that you will find it highly cost-effective to acquire additional training materials for areas of CAD/CAM/CNC which are new to you. Tormach sales can help point you in appropriate directions.

2.2 Receiving, Unpacking and Checking Shipment

2.2.1 Moving the Crate

The PCNC 1100 is supplied on a standard pallet and can be offloaded from a truck with a tailgate lift and moved on smooth surfaces using a hydraulic pallet jack. This makes delivery very economical (figure 2.3).

Remove the crate top and sides with care as the axis drive stepper motors are in vulnerable places (figure 2.4).



The crated system weighs less than 1300 lbs (600 kg) nevertheless, it requires mechanical handling to move it over rough ground and to lift it onto the stand. Tormach advises you to employ the services of a specialist rigger if the machine has to be moved in situations where the pallet lifter cannot be used or where there is no crane to lift the machine onto its stand. It is possible to improvise using a small trailer, a portable engine crane and similar tools if there is no alternative but this risks injury to you and damage to the machine.



Figure 2.3 – Crated machine as delivered

2.2.2 Uncrating and Inspection

After uncrating you should check the contents against the parts listed on the packing slip and inspect the machine for any damage incurred during transit so any claims can be made within the carrier's deadline.



Figure 2.4 – Un-crated machine on pallet

2.3 Assembling Y-axis Drive

The PCNC is supplied with the Y-axis drive motor mechanically disconnected. You are advised to mount it before attempting to remove the machine from its pallet.

- Un-strap the stepper motor from its transit position (figure 2.5).
- Remove the sheet steel cover from the Y-axis drive coupling box (at base of column behind the machine).
- Remove the four cap head screws from the back face of the coupling box and loosen the two screws in the shaft coupling on the end of the Y- axis ballscrew (4 mm or 5/32" hex wrench).
- Use the cap screws to mount the motor on the back face of the coupling box. The wiring runs downwards from the motor. It is important to ensure that the motor flange can pull up flush to the machined face of the coupling box. Remove any paint that could cause mis-alignment. After tightening the cap screws, back them off ¼ turn so the motor is free to self align.



Figure 2.5 – Y-axis motor as shipped and mounting flange



Preparation

- Ensure that the coupling is centrally positioned on motor shaft and the machined end of the ballscrew and tighten the cap screws on the coupling. Then tighten the cap screws holding the motor (figure 2.6).
- Check that the axis turns with no tight spots. The smooth (i.e., outside face) of an old auto engine timing belt or poly-vee auxiliary drive belt, which has been cut to make a strip, can be used to turn the coupling between the stepper motor and ball screw (figure 2.9). If it does feel tight then you need to recheck the alignment of the coupling.
- A final check of alignment should be made when the machine is under computer control. This involves loosening the motor retaining screws $\frac{1}{4}$ turn and jogging the axis. The motor should show no signs of wobbling. If it does wobble or move relative to the coupling housing then the coupling screws should be slackened and retightened and the motor rechecked for movement. When alignment is perfect, retighten the motor fixing screws and refit the coupling box cover.

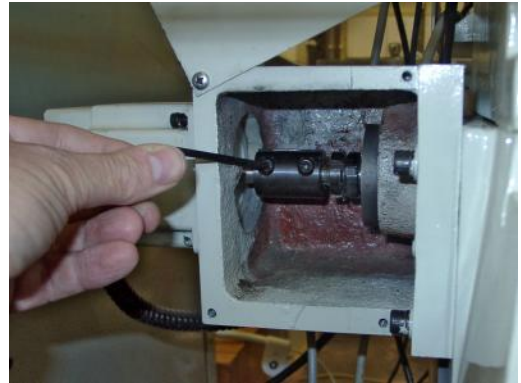


Figure 2.6 – coupling the Y drive motor

2.4 Mounting the PCNC

2.4.1 Lifting onto Stand

The machine can be lifted onto an operating stand by either of two methods: from below using the base connection points or from above using a slinging technique. In either case caution and common sense are needed for the protection of the machine and the people involved. Lifting up to 1000 lbs can be simple with proper preparation and good equipment, but it is never trivial and the dangers involved should be taken seriously.

The work of lifting and placing heavy equipment is called rigging. If you are not trained or prepared then you should seek the advice of those who are. Professional riggers can be found in most areas.

2.4.1.1 Lifting from Below

The base of the machine has four $\frac{7}{8}$ " holes. By sliding two steel bars into these holes, at least 32" in length, you end up with some outrigger wings that can be used in combination with a fork lift truck to lift the machine. These should be solid steel bars, not pipes and be $\frac{3}{4}$ " or $\frac{7}{8}$ " in diameter.

2.4.1.2 Lifting from Above

The alternative way to mount your PCNC to a stand involves lifting from above. The eye in the top of the column is suitable for lifting the machine, but it is not in line with the center of gravity. The machine will tilt when lifted solely from the eye. The alternative is to sling the machine using a combination of the eye and an eye in a T-nut on the table using a Tormach special tool (part number 30576 - Machine Hoist Bar) (figure 2.7).



Figure 2.7 – Hoist bar for slinging the mill

Preparation

Figure 2.8 shows the geometry of the slinging. The table should be as far away from the column and as far to the right as possible to optimize the balance.

It is most important that the machine is not lifted by the control cabinet or by any of the protruding stepper motors or the head or the table. Incorrect rigging of the sling will likely result in serious damage to the PCNC 1100.

The optimal balance for lifting should be checked with the machine an inch or two off the floor.

The X-, Y- and Z-axes can easily be moved by hand if the covers on their coupling boxes are removed. The smooth (i.e., outside face) of an old auto engine timing belt or poly-vee auxiliary drive belt, which has been cut to make a strip, can be used to turn the coupling between the stepper motor and ball screw (figure 2.9). As an alternative to adjust the Y-axis to the front position before you fit the stepper motor, you can temporarily clamp a length of $\frac{1}{2}$ " bar into the coupling and turn the bar with a pair of slip-joint pliers (figure 2.10).

When you are ready to lift the machine you should remove the nuts from the four screws holding it down to the pallet.

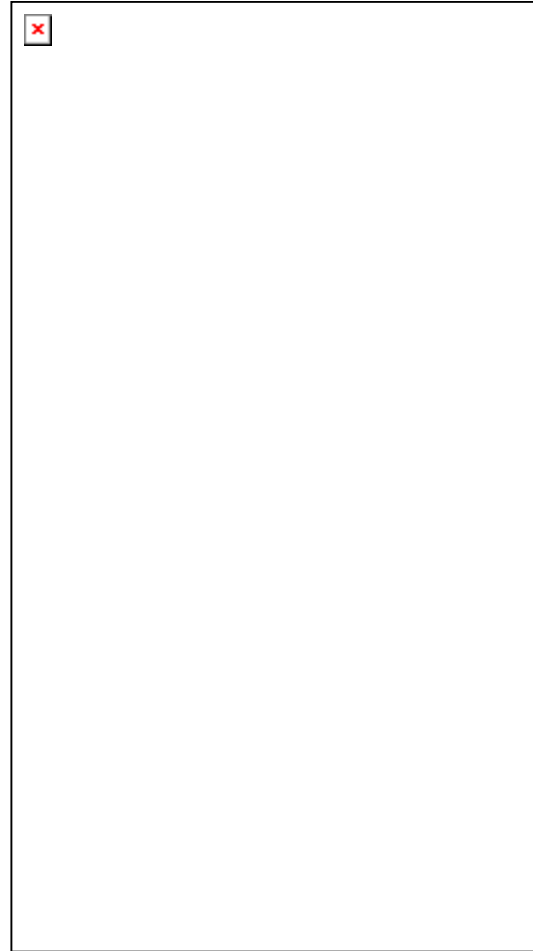


Figure 2.8 – Slinging geometry

2.4.2 Fixing to Stand

The surface to which the machine is bolted must be flat. Firmly bolting the machine to a surface that is not flat will stress the frame and must be avoided. If necessary, thin shims should be inserted at the holding down points so that bolting it down to the stand cannot distort the base. This is important to preserve the accuracy of the PCNC1100.



Figure 2.9 – Manual moving table by back of old toothed belt

2.4.3 Accessories

You should now try out the positioning of the screen, computer and coolant sub-system, if any.

2.5 Power to the PCNC

The PCNC 1100 is powered by 230 volt single phase AC (50 or 60 Hz). Auxiliary services like the control computer and coolant are separate and can be 115 or 230

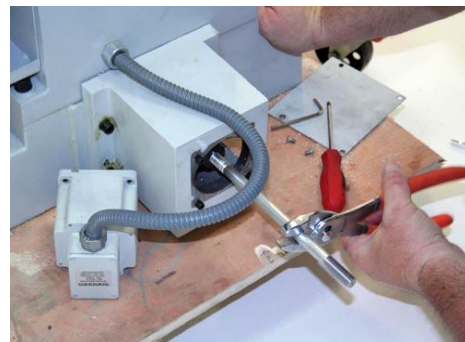


Figure 2.10 – Manual positioning of Y-axis to balance machine

volts.

The main machine power lead is shipped in a protective tray. Terminate it with a suitable plug for the wall outlet which you intend to use.

2.6 Power for Machine Accessories

There is an IEC inlet on the bottom of the control cabinet for the computer/coolant. This should be connected to a GFI (Ground Fault Interrupt) wall outlet. This separate supply, like the main machine supply, is controlled by the **main power switch** but allows use of 115 volt accessories on a 230 volt mill.

The 115 VAC outlet under the panel that is furthest from the column feeds the coolant pump and is controlled by the CNC software. The pair of similar outlets nearer the column is switched by the **Computer** switch on the front panel.

Both the 115 VAC power and ground are autonomous from the main machine power and ground (230 VAC) in order to allow correct operation of the ground fault interrupter (GFI). The computer control of the coolant outlet is accomplished by an isolated relay. Refer to the upper portion of the circuit diagram in section 10.2 for details, wire numbers 200 through 207.

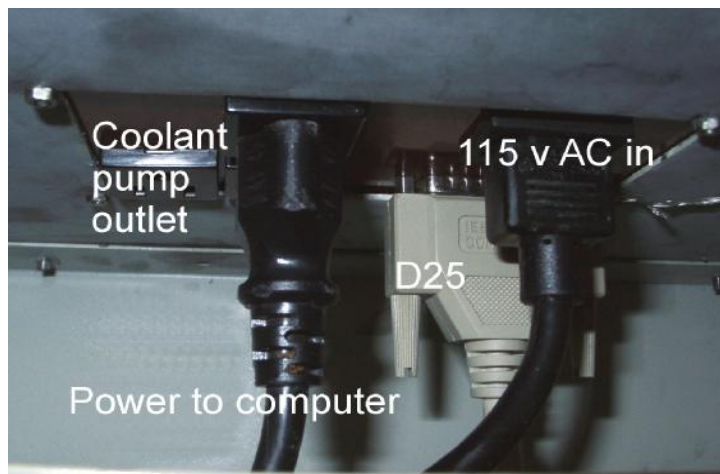


Figure 2.11 – Power and interface connectors



2.7 Control Computer and Software Installation

2.7.1 Computer Specification

You need to provide a computer running Microsoft Windows XP (Home or Professional edition). Consult Tormach LLC for information on use of the PCNC with Windows Vista.

The system does not require a particularly powerful computer but there are certain requirements for optimal operation. You should have:

- A CPU running at least 1 GHz with at least 256 megabytes of RAM. Intel and AMD processors are both satisfactory.
- Video of 1024 x 768. Certain low cost computers with integrated video have shown problems. In those situations the best solution is to disable the motherboard based video and install an inexpensive video card in an expansion slot.
- Most laptop computers are unsuitable and any which can be made to work will only do so by re-installing Windows without the APCI (Advanced Power Configuration Interface). This will cripple the machine for normal portable use.
- You need a parallel printer port (25 pin D socket). A USB to printer adaptor cannot be used in place of this. If this port is on the motherboard then the standard configuration will use it. PCI parallel ports from many manufacturers can, however, be used. You will need to consult Chapter 9 for details of the configuration process for cards which do not use the standard port address of 0x378.
- Some motherboards based on the Via chipset have proved incompatible with the control software. If you are buying such a board please ensure your supplier will take it back and refund the cost in the unlikely event that you have problems.

Preparation

- A CD ROM drive is essential for software installation.
- We advise you to use a USB storage device for transporting small files like G-code programs to the machine tool. These devices are variously known as “flash,” “thumb,” “jump” or “key” drives. The most common U.S. usage is “flash drive” and we will use that from now on. Please note, G-code programs should not, however, be run directly from the USB storage device. Transfer the G-code programs to the local hard disk and run them off the hard disk, not off the flash drive.
- The computer can have network capability but it should not be used while the machine tool is being used. In particular, Ethernet cards which will auto-negotiate to work at 10BaseT or 100BaseT should be set for a fixed speed or you will hear a “tick” from the steppers every second or so during jogging as Windows takes control to negotiate network speed. Do not run G-code programs off a network drive or remote computer.
- Do not allow background automatic updates to be active while running the mill. Do not leave Firefox, Mozilla, Internet Explorer or Netscape run in the background while operating the mill.

There are some features in the computer that will make operation of the PCNC more convenient. Things to look for are:

- A computer which will boot-up when it sees AC power switched on – rather than needing you to press a button. This feature generally is controlled by a BIOS option and can be enabled by entering the BIOS configuration mode.
- A keyboard that includes a small USB hub. This will make it easy to plug in the pointing device (mouse/track-ball), a pendant and a USB flash drive.
- If you choose to use a mouse or trackball then, you should get the “optical” kind. Mouse balls and workshop conditions do not mix well.

2.7.2 *Optimizing the Windows Installation*

It is important that the computer used to control your PCNC does not have a large load of software running in the background. Examples are the automatic update utilities for Windows and virus/spam checkers, Multimedia software and telephony and messaging software. Such software can wake up at any time and place large loads on the computer and its disk subsystem. This can sometimes interfere with timing of movement of PCNC and lead to spoiled work.

The simplest advice is:

- If you are using an existing computer that has been used for some time there are likely to be a number of unnecessary programs or drivers loaded when the computer boots. Windows can be hard to “clean up,” the best solution is often to re-install Windows after formatting the hard drive. You may prefer to ask an IT specialist to help you with this.
- Install the minimum options for Windows. You should include access to the Internet so you can maintain your system software and access the Tormach web site but you will normally run with the network disabled.
- Do not install other software packages on the computer. If you wish the control computer to be dual purpose then we advise you to create two partitions on the hard drive. Install Windows in both but keep one as the minimal installation for the control software for the PCNC. The other partition can contain what you like. When you switch on you can choose from which partition you want to boot.

Appendix 3 gives a checklist of actions to achieve the “leanest” Windows system. The recommendations there are helpful, but not always necessary. Do not leap in and do everything unless you have problems with machine motion. Apply them in the order given until jogging and rapid moves are smooth. Optimization of Windows is more important on slower computers (below 1.6 GHz).

2.7.3 Installing the Control Software

2.7.3.1 Installing

You do not need the PCNC connected to the computer by the parallel cable yet. If you are just starting it would be better not to have it connected. You **must not** have a printer connected to the parallel port. The Control Software will not operate correctly with a printer attached to the parallel port and it may damage the printer. Switch off the PC, the PCNC and unplug the 25 pin connector from the back of the PC. Now switch the PC back on.

Load the release CD into your CD drive. If you have “auto run” configured on your computer then the installer will start running.

Otherwise use Windows Explorer or the My Computer icon to open the CD. Select View>Details from the menu. Double-click on the file Install.BAT to run it. You will be guided through the usual installation steps for a Windows program such as accepting the license conditions and selecting the folder for the control software. Accept all the defaults. You will now be told to reboot before running the Control Software.

This installation process puts a copy of your numbered license file and the working license into the PCNC folder.

2.7.3.2 Vital Re-boot

After you have initially installed the Control Software program, you must re-boot your computer. This reboot is **vital**. If you do not do it then you will get into great difficulties which can only be overcome by using the Windows Control Panel to uninstall the driver manually. **So please reboot now.**

If you are interested in knowing why the reboot is required then read on, otherwise skip to the next section.

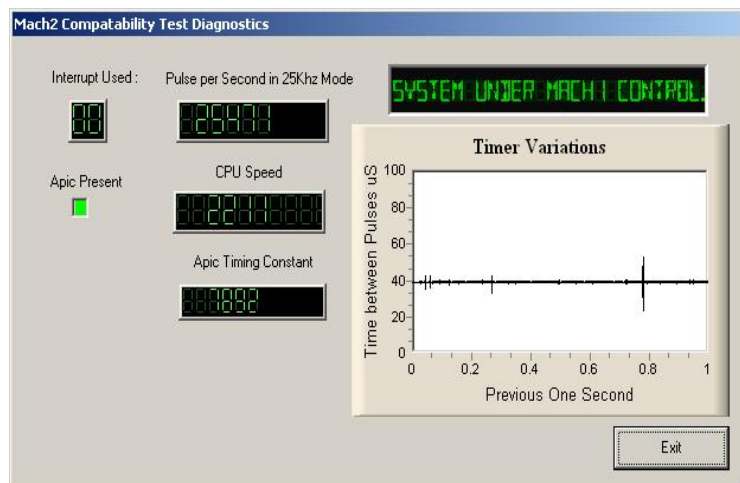


Figure 2.12 – The DriverTest program display

Although the Control Software will appear to be a single program when you are using it, it actually consists of two parts; a driver which is installed as part of Windows like a printer or network driver and a graphical user interface (GUI). The reasons for this division are complex but the driver is the most important and ingenious part.

The Control Software must be able to send very accurately timed signals to control the axes of the machine tool. Windows likes to be in charge and runs normal user programs when it has nothing better to do itself. So the Control Software cannot be a “normal user program;” it must be at the lowest level (highest priority in the computer) inside Windows (that is, it handles interrupts). Furthermore to do this at the high speeds possibly required (each axis is given attention 25,000 times per second) the driver needs to tune its own code. Windows does not approve of this (viruses often play this trick) so it has to be asked to give special permission. This process requires the reboot. So if you have not done the re-boot then Windows will give the *Blue Screen of Death* and the driver will be corrupt. The only way out of this will be to manually remove the driver.

Having given these dire warnings, it is only fair to say that, although requested when upgrades are installed, the reboot is only strictly required when the driver is first installed. Windows XP boots reasonably quickly that it is not much hardship to do it every time.

2.7.3.3 Testing the Installation

It is now highly recommended to test the system. The Control Software is not a simple program. It takes great liberties with Windows in order to perform its job in fact it actually runs Windows rather than Windows running it; this means it will not work on all systems due to many factors. For example, QuickTime's system monitor (qtask.exe) running in the background can kill it and there will be other programs which you probably are not even aware are on your system that can do the same. Windows can and does start many processes in the background; some appear as icons in the systray and others do not show themselves in any way. Other possible sources of erratic operation are local area network connections which may be configured to automatically speed detect. You should configure these to the actual speed 10 Mbps or 100 Mbps of your network.

Because of these factors, it is important that you test your system when you suspect something is wrong or you just want to check that the install went well.

Navigate by Windows Explorer to the PCNC folder (usually C:\PCNC3). Double click the file DriverTest or if you display file extensions DriverTest.exe (figure 2.12).

You can ignore all the boxes with the exception of the Pulse Frequency. It should be fairly steady around 24,600Hz, but may vary, even wildly, on some systems. This does not necessarily mean the pulse timer is unsteady, it may mean that the computer is heavily loaded or slow to begin with, since the Control Software takes the highest priority in the system, the clock may be shunted down to a priority slow enough that its one second is a variable length of time. Since the pulse count is based on one second of Windows time, variations in Windows time will make the pulse count look like it is swinging around a lot even when it is rock solid. Basically, if you see a similar screen to figure 2.12, everything is working well so **close the DriverTest program and skip to the section *Running the PCNC* below.**

Windows "experts" might be interested to see a few other things. The white rectangular window is a type of timing analyzer. When it is running it displays a line with small variations indicated. These variations are the changes in timing from one interrupt cycle to another. There should be no lines longer than $\frac{3}{8}$ " or so on an 17" screen on most systems. Even if there are variations its possible they are below the threshold necessary to create timing jitters so when your machine tool is connected you should perform a movement test to see if jogging and G00/G01 moves are smooth.

You may have one of three things happen to you when running the test which may indicate a problem.

1. Screen shows wide variation or clearly periodic variation of timing. In this case you need to complete a computer optimization detailed in Appendix 3 (section 10.3 of this manual).
2. "Driver not found or installed, contact Art." This means that the driver is not loaded into Windows for some reason. This can occur on XP systems which have a corruption of their driver database, reloading Windows is the cure in this case.
3. When the system says, taking over...3...2...1... and then reboots, one of two things has occurred. Either you did not reboot when asked (told you!!) or the driver is corrupted or unable to be used in your system. In this case follow the next section and remove the driver manually, then re-install. If the same thing happens, please notify Tormach by e-mail and you will be given guidance. A few systems have motherboards which have hardware for an APIC timer but whose BIOS code does not use it. This will confuse the Control Software install. A DOS batch file "specialdriver.bat" to run in a DOS window is available; this will make the driver use the older i8529 interrupt controller. To display the DOS window enter CMD as the program after Start Button>Run. You will need to repeat this process whenever you download an upgraded version of the Control Software as installing the new version will replace the special driver.

2.7.3.4 DriverTest After a Software Crash

Should you for any reason have a situation when the Control Software crashes – this might be an intermittent hardware problem or software bug – then you must run DriverTest as soon as possible after the Control Software has failed. If you delay for two minutes then the Control Software driver will cause Windows to fail with the usual *Blue Screen of Death*. Running DriverTest resets the driver to a stable condition even if the rest of the Control Software disappears unexpectedly. If you have a troublesome system then you might find it worthwhile to create a shortcut for DiverTest.exe and drag it onto the Desktop for easy access.

2.7.3.5 Manual Driver Installation and Un-installation

You only need to read and do this section if you have not successfully run the DriverTest program.

The driver (Mach3.sys) can be installed and uninstalled manually using the Windows control panel.

- Open the Windows Control Panel and double-click on the icon or line for System.
- Select Hardware and click Add Hardware wizard. As mentioned before the Control Software's driver works at the lowest level in Windows. Windows will look for any new actual hardware (and find none).
- Tell the wizard you have already installed it and then proceed to the next screen.
- You will be shown a list of hardware. Scroll to the bottom of this and select Add a new hardware device and move to the next screen.
- On the next screen you do not want Windows to search for the driver so select Install the hardware that I manually select from a list (Advanced).
- The list you are shown will include an entry for Mach x pulsing engine. Select this and go to the next screen.
- Click Have disk and on the next screen point the file selector to your directory (C:\PCNC3 by default). Windows should find the file Mach3.inf. Select this file and click Open. Windows will install the driver.

The driver can be uninstalled rather more simply.

- Open the Control panel and double-click on the icon or line for System.
- Select Hardware and click Device Manager.
- You will be shown a list of devices and their drivers. Mach x Pulsing *Engine* has the driver *Mach3 Driver* under it. Use the + to expand the tree if necessary. Right-click on Mach3 Driver gives the option to uninstall it. This will remove the file Mach3.sys from the Windows folder. The copy in the PCNC3 folder will still be there.

2.8 Running the PCNC

You have now completed the installation and merely need to connect the PCNC to the computer.

Now close down Windows and switch off the PC. Connect the parallel port of the computer to the D25 connector on the underside of the PCNC control cabinet. The cable provided to connect the computer to the mill meets IEEE 1284 specifications. This provides a high level of immunity to electrical noise, which is important to reliable operation. Do not use inferior cables. Power the computer from the outlet near the D25 connector and if relevant, power the monitor from the connector adjacent to it below the PCNC control cabinet. Viewed from behind, the inlet is on the right next to two outlets for computer and monitor. The coolant pump outlet is to the left of these (figure 2.11).

2.8.1 Main Switch and Control Panel

The rotary **main switch** on the right hand side of the control cabinet disconnects the mains power from the PCNC itself and isolates computer/coolant (115/230 VAC) outlets that are on the bottom of the cabinet.

Warning: You should not open the control cabinet until the mains power to the machine and to the computer/coolant pump are both removed from the wall outlet. Live parts may be exposed even when the main switch is in the off position.

Computer On/Off

Switch the **main switch** ON and switch the computer power on at the PCNC control panel (figure 2.13).



Figure 2.13 – Control panel layout

If your computer does not power up when power is applied to the cord you will have to press the computer's power-on button. Restart the computer and use the desktop shortcut (PCNC-M3) to run the Control Program (figure 2.14).

This will allow you to perform all the important functions on the standard machine. If you have the 4th axis or want to use G-code features like Optional Stop then you may wish to use the Comprehensive Run screen (figure 2.15). In this case click the Comp Run button to change the display.

The screen "Light Emitting Diodes" (LEDs in this manual) by the Reset button will be flashing as will the Machine OK LED.

Now, returning to the operator's panel:



Figure 2.14 – Simple main screen

Start and EStop

The **Start** button will energize the circuits for the axis drives and for the spindle motor. The **Stop** button stops all motion and is the Emergency Stop (EStop) control. The **Stop** button locks in the off position once it has been pressed as safety feature. It can be released by a turning the button-head a quarter-turn clockwise. **Note:** Once the stop button has been pressed the start button is inoperative until the stop button is released.

The **Machine** LED indicates that the **Start** button has been pressed. When it is lit then the Machine OK LED on the computer screen should be solid green. If this does not happen then you should check that the cable between the PCNC and computer is fully plugged-in at both ends. You will be able to test some controls on the PCNC, even if the LED does not give the correct indication but you will have to find the fault before you can move the PCNC axes under computer control.

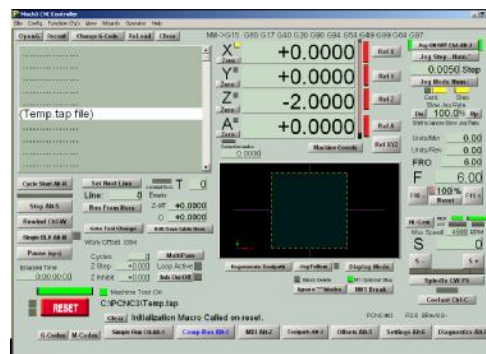


Figure 2.15 – Comprehensive Run screen

Spindle

An important safety interlock is that the rotation of the spindle can be disabled (while maintaining axis drive power) by the **Spindle Lockout** key-switch. The switch on the spindle drive door performs the same function. These are used to ensure that the spindle cannot start when an R8 tool holder is being changed (door open) or when a tool is being changed in a collet chuck. Turning the **Spindle Lockout** key or opening the spindle cover while the spindle is running will also stop the spindle. Stopping the spindle in that way will not damage any components, but it is generally a poor practice.

The controls to the right of the key-switch are all concerned with manual control of the spindle. If the **Manual/Auto** switch is in the **Auto** position then none of the other spindle controls have any effect. In the **Manual** position the computer control of the spindle is disabled.

Switch to **Manual** and turn the speed control knob fully counterclockwise. Press the **Spindle Start** rocker. The spindle should start turning slowly in the clockwise (forward) or counterclockwise (Reverse) direction (viewed from above) depending on the setting of the **Forward/Reverse** switch. You can safely switch directions while the spindle is turning.

Try changing the speed using the rotary control knob. The actual speed will depend upon which of the two pulley ratios you have selected.

Pressing the **Spindle Stop** rocker will halt the spindle.

Coolant

The **Coolant** switch controls the power to the coolant pump outlet on the underside of the control cabinet. In the **Off** position the outlet is not powered and the pump will not run. In the **On** position power is applied to the outlet and the pump will run until the switch is switched to either off or auto positions. In the **Auto** position outlet power is under program control the pump will run if the Control Program requests coolant.

Accessory Socket

The Accessory socket is for connection of accessories such as a touch probe. See Chapter 8 for details of the interface.

2.8.2 Changing the Spindle Speed Range

The PCNC has two speed ranges. The low range, 350 to 1750 RPM, is suitable for most machining operations with ferrous and other tough materials. The high range, 900 to 4500 RPM is suitable for small diameter cutters, plastics and non-ferrous materials. The range change is performed by moving the V-belt from the upper pair of pulleys (high speed range) to the lower pair (low speed range).

Open the spindle drive door. The interlock will prevent the motor from running. **Note:** Opening the spindle drive door will also stop a running spindle; however, this is a safety hazard and should not be used as a substitute for stopping the spindle with the spindle controls in the Control Software. Use the rear handle to unlock the motor mounting plate and pull the motor forwards. The belt will slacken and can be moved from one set of pulleys to the other (figure 2.16).



Figure 2.16 – Changing spindle pulleys

For the following tests, select the low speed range by placing the belt on the lower pair of pulleys. Retighten the belt so there is between $\frac{1}{8}$ " and $\frac{1}{4}$ " movement between the pulleys, lock the motor mounting and stow the handles in the vertical position.

2.8.3 Computer Control of the Spindle and Coolant

Switch the **spindle** and **Coolant** to **Auto** on the control panel. Make sure the computer is displaying the Simple Run screen as shown in figure 2.14. The portion shown in figure 2.17 shows the controls for the spindle.

Use the mouse to click the Hi/Lo button. You will see that the screen LEDs depicting the pulleys will change and the appropriate maximum speed will be displayed below them. Choose the Low setting to correspond to the PCNC pulleys.

Note: Just after starting the system the screen LEDs may not correspond to the indicated maximum speed. The speed value is always correct and a click on the Hi/Lo button will bring the LEDs into step.

Next to the label S, is a digital read-out (DRO) of the requested spindle speed. You can change this by clicking the mouse on it. It will become highlighted. Type a number, say 500 (for 500 rpm) and press *Enter*. Figure 2.18 shows the screen just before pressing *Enter*. If you make a mistake you can press *Esc* to return to the original value.

This technique is used for setting any DRO. Remember to use *Enter* after any DRO change. If you forget and just click on another DRO, then any value you have just entered will be discarded. This is designed to avoid accidental changes.

Now check that the machine is safe and that the motor door is closed and click on the Spindle CW F5 button. The spindle will start running. Clicking the button again will stop it.

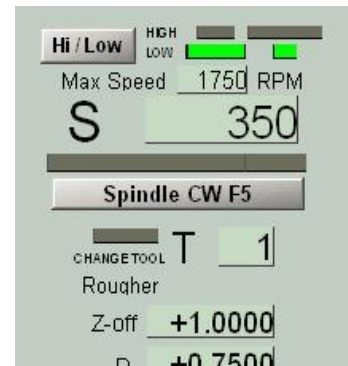


Figure 2.17 – Spindle controls



Figure 2.18 – Setting S word

Preparation

The F5 in the caption tells you that function key *F5* is a “shortcut” to this button; it can also be used to start and stop the spindle.

Notice that there are two sets of screen LEDs. The outer set indicates that the machine is dwelling to ensure the spindle has started and fully stopped. The center LED indicates that it is running.

If you have connected a coolant pump to the outlet under the control box (see vacant outlet in figure 2.10) then you will be able to control it by the Coolant Ctrl-C button or its shortcut which is the *Ctrl-C* key. Beware of the position of the coolant nozzle before you try this!

2.8.4 MDI for Entering G- and M-code Commands

When you are making parts the commands to the machine (G- and M-codes) will generally be read from a file. It is however often convenient to command the PCNC directly. This can be done by typing command into the Manual Data Input (or MDI for short) line.

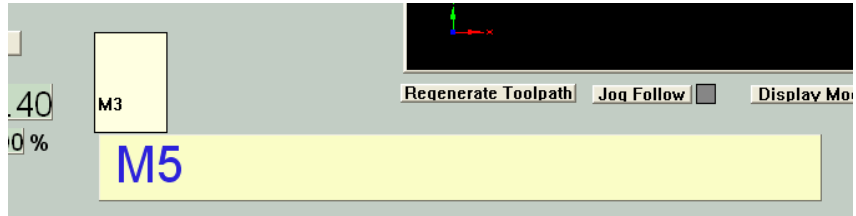


Figure 2.19 – MDI Line in use

The command to start the spindle in the clockwise direction is M3 and the command to stop it is M5.

Click the mouse in the bar marked MDI. It will highlight. You type the command in the highlighted line. Unlike in DROs, the *Backspace*, *Del*, *Left* and *Right* arrow keys are available to help you correct any typing errors.

When you press *Enter* the command will be executed. Pressing *Esc* abandons it and closes the MDI line. You can try starting and stopping the spindle with M3 and M5 G-codes.

Notice that the recent commands are displayed in a fly-out box. You can choose one of these to copy into the MDI line using the *Up* and *Down* arrow keys.

Figure 2.19 shows the MDI line after the spindle has been started (M3) and the M5 has been typed but not yet executed by *Enter*.

There are some handy features of the MDI box. It can be opened by pressing *Enter* (rather than needing a mouse click). It stays open after a command has been executed. It can be closed by *Enter* when it is blank.

Note: All keystrokes go to the MDI when it is open so it is not possible to execute shortcuts or jog the axes. If the keyboard does not do what you expect then you probably still have the MDI line open.

2.8.5 Jogging the Axes

The final thing to try before actually making your first part is to move (jog) the PCNC axes using the keyboard.

There are several options for jogging which will be explained in detail below. The jogging controls are at the top right hand side of the screen (figure 2.20).

Click the Jog ON/OFF button (or use its shortcut *Ctrl-Alt-J*) to turn on the screen LEDs beside it if they are not already on. Click Jog Mode (or use its shortcut) to turn on the larger of the LEDs above the word Cont. (for

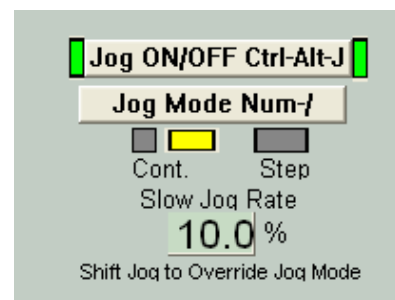


Figure 2.20 – Simple jog controls

Preparation

Continuous jogging). Type the value 10 into the Slow Jog Rate DRO; do not forget the *Enter* to accept the value)

In the next steps you may find the directions of movement are unexpected. Therefore, when you use the arrow keys you should be prepared to quickly release the key if the axes moves in an unexpected direction or is near its limit of travel. Now, press the Left, Right, Up and Down arrows on the keyboard. The table will move while you hold the key down. If you crash into the limit switches then the LED beside the red *Reset* button will flash and the machine will stop. Click on Reset and very carefully jog the other way. Take care not to mechanically hit the limit doing this in the wrong direction. You will lose the referenced status if you trip a limit switch.

You might find the directions of movement are unexpected. Figure 2.21 shows the tool above a work piece. The positive X, Y and Z directions are marked by arrows. If you press the *Right* arrow key then the **tool** will move in the positive X direction (i.e., to the right of the work). Similarly the *Up* arrow moves the tool in the positive Y direction (i.e., towards the PCNC column). Of course, this actually happens on the PCNC by the table moving under the tool but you must imagine what it would look like if you were sitting on the table and watching the tool.

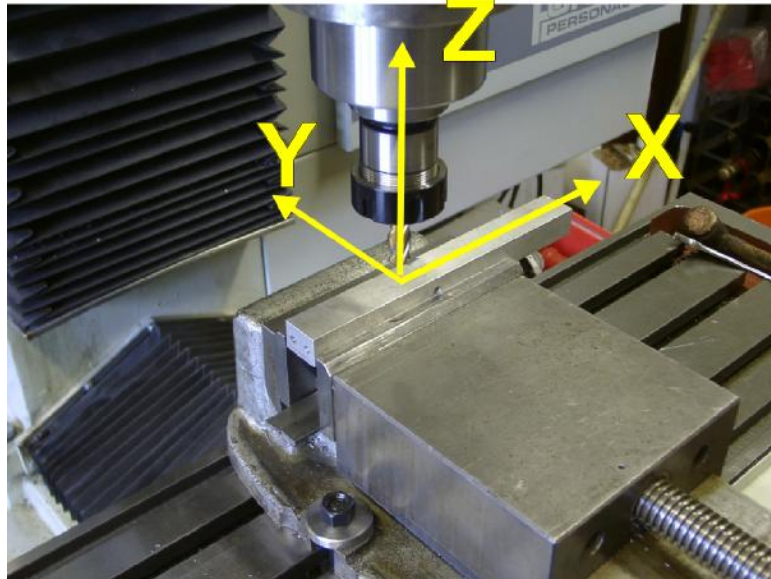


Figure 2.21 – Jogging is to move the tool relative to work

You should jog around until you are quite confident which way the machine moves when you press any key. Notice that the values in the Axis DROs change as you jog the tool around.

Jogging the Z-axis is done by using the *Page up* and *Page down* keys. Here of course it is the tool that moves, so “Up” is indeed up.

2.9 Summary

This chapter has covered a lot of basic ground. Much of it only has to be done once. You may however wish to revisit the latter parts if you are not fully confident with using buttons, DROs, the MDI and jogging. We will give less detail on using these (e.g., assume you know about shortcuts and when to use *Enter*) in subsequent chapters.

3. Making Your First Part

This chapter shows you how to make your first part with the PCNC. It assumes that you have no prior experience with running a part-program on a Computer Numerically Controlled machine tool.

Even if you have previous experience you will find that following this tutorial gives you an introduction to the controls of the machine.

3.1 Coordinates

Coordinates are simply how you describe where the tool is positioned. We assume that you have the computer and PCNC setup and switched on and are looking at the Simple screen.

3.1.1 Referencing the Machine

You have seen that jogging the machine moves the tool and this changes the numbers in the X-, Y- and Z-axis DROs. You may have wondered where these numbers come from and indeed doubted that they are very sensible. The answer is that they are probably meaningless. The computer has no way of knowing yet where the tool or table is positioned.

Referencing is the process which puts the machine in a known mechanical position and sets the corresponding axis DROs. The known position is where the limit switches operate with Z at the top of the column and the spindle over the bottom left-hand corner of the table. This position is called Home and so these limit switches are called the Home switches. Their operation is very obvious on the Z-axis. A software trick makes the X and Y arrangement slightly harder to understand but this is of no importance at present.

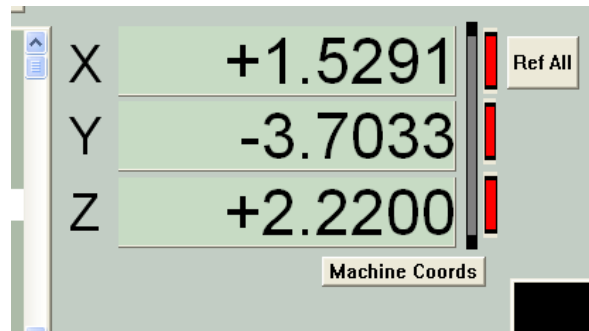


Figure 3.1 – The axis DROS un-referenced

Figure 3.1 shows how the DROs might look when you start up the system. They have arbitrary values in them and the three screen LEDs are red.

Check that the LEDs by the red Reset button are not flashing and that nothing is in the way of movement of the table and head of the PCNC. Click the Ref All button.

The Z-axis will first move up to the top of the column, stop at the switch and then move down a fraction. The Z DRO will be set to zero and its LED turn to green. This will be followed by the same action with X and Y. Be wary the first time you do this and if an axis does not stop at its switch you must hit the **Estop** button on the PCNC control panel and look at the Maintenance/Troubleshooting section of this manual (figure 3.2).

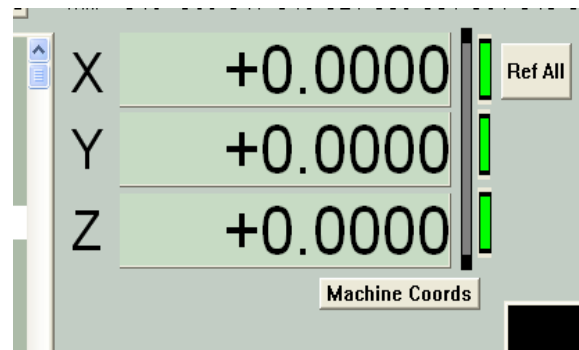


Figure 3.2 – Referenced and ready to use

Note: It is very important to reference the PCNC before using it. Failure to do so can result in running into the limit switches or, worse, the tool attempting to cut into the vise or table. The Control Program tries to protect you from this sort of trouble but intentionally leaves the responsibility of referencing to you. You can repeat the referencing operation at any time if you have reason to suspect that the PCNC is in the wrong position (e.g., an error in a depth of cut has stalled the machine).

The Control Software now knows exactly where the PCNC axes are and has called this home position $X = 0.0$, $Y = 0.0$ and $Z = 0.0$. You probably think that this position is not very convenient and you are correct. The Control Software will let you define any other place to be 0, 0, 0 when you are running a part-program but will always keep a record of the machine position using home as machine zero. Normally the axis DROs will show **your** coordinates for the position but you can see the Control Software **machine** coordinates by clicking the Machine Coords button. A big flashing LED warns you that you are not looking at your version of the coordinates.

3.2 Loading a Demonstration Program

The system comes with a demonstration program in the file `C:\PCNC3\GCode\FirstPart.nc`

Run the Control Software and make sure the screen LEDs by Reset button are not flashing and that the axes are referenced.

Click the OpenG button. You will be given a Windows open file dialog. Navigate to the G-code folder and open the file (figure 3.3).

You will see the code of the program in the window at the left of the screen and the path that the tool will take in the toolpath display. Use the mouse to drag with the left-hand button in the toolpath display to rotate the display. It behaves as if it is drawn inside a clear ball and the mouse turns the ball.

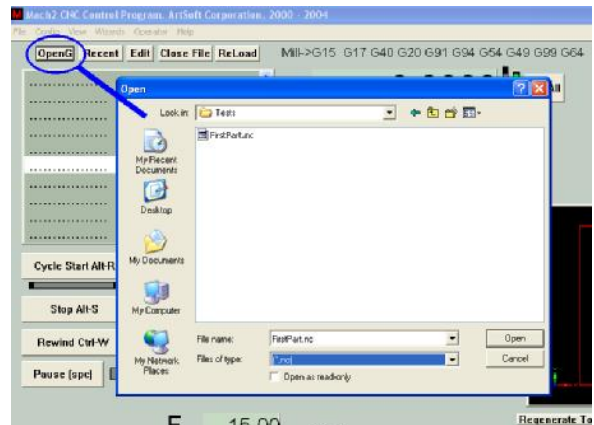


Figure 3.3 – File opening

So that you understand what is going to happen we should look at the way this file was produced using a Computer Aided Design/ Computer Aided Manufacturing (CAD/CAM) program. The PCNC will produce parts designed in any available industry standard tools. The CAD/CAM system (Turbo CAD/CAM 3) used for examples in this manual is one such tool that is very powerful but reasonably priced.

The original drawing is shown in figure 3.4. It consists of the characters “PCNC” and an irregular quadrilateral.

Having drawn the outline of the part, we defined the material from which it will be cut, the size of tool to cut the letters ($\frac{1}{4}$ ”), the depth of the letters, the size of tool to cut the recess ($\frac{1}{2}$ ”) and the depth of the recess. The program automatically creates the part-program (G- and M-codes) to make the

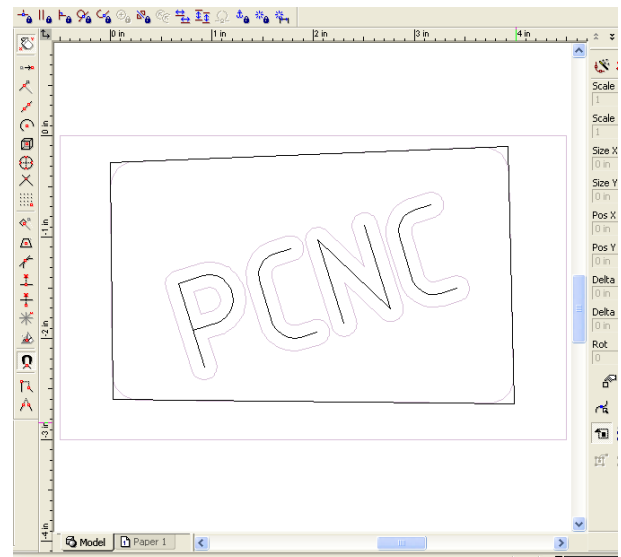


Figure 3.4 – The drawing for FirstPart

part and can display a visualization of the finished work (figure 3.5). We will discuss the options which you have for designing your own parts in the next chapter.

You should now be able to relate the toolpath display to the part illustrated

3.3 Running the Demonstration Part-program

3.3.1 Part Material

We suggest that you use a free-cutting material to make this first part. MDF, birch ply or, as we used, resin impregnated ply are all suitable. You need a piece at least ½" thick and 7" by 4". Clamp this securely to the PCNC table. For many jobs it is worth putting a layer of scrap material under the work. This will protect the table when through holes are to be cut and may save damage in the event of a mistake in the part program or in operating the machine.

Put a ¼" diameter tool into a tool holder in the spindle. Ensure the lower spindle pulleys are selected on the machine and are set on the screen using the Hi/Lo button.

Use masking tape to indicate a 5" length near the middle. This length must be clear of the clamps.

3.3.2 Setting Work Offsets

Jog the axes so that the tool is just above the material at the top left-hand corner of the 5" section (figure 3.6).

We suggest that you do a dummy run cutting in the air so jog the Z-axis up about two inches.

Now click in the X-, Y- and Z-axis DROs, in turn and type the value 0 into each of them. Do not forget to press Enter to accept the values. The reason for doing this is that you want the position of the tool (i.e., at the top left of your material and an inch above it) to be X = 0, Y = 0 and Z = 0. The Control Software still refers to the home switches as zero internally but lets you have your own separate coordinate system for the work. If you click the Machine Coords button then you will be shown how your zero relates to the home position. The difference between the machine

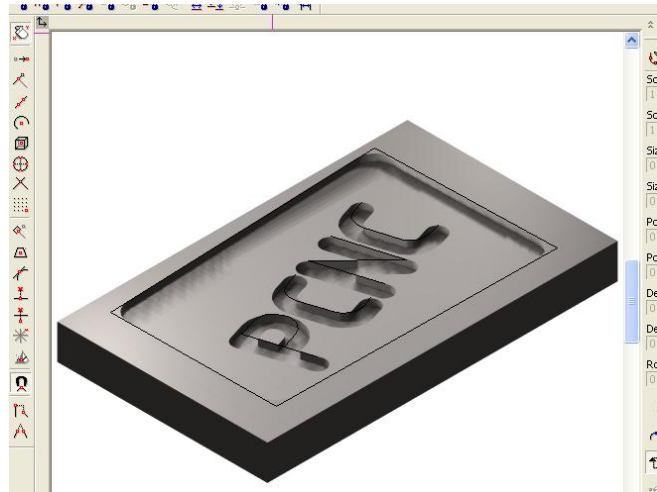


Figure 3.5 – CAD/CAM Visualization of finished part



Figure 3.6 – Tool setup at zero points

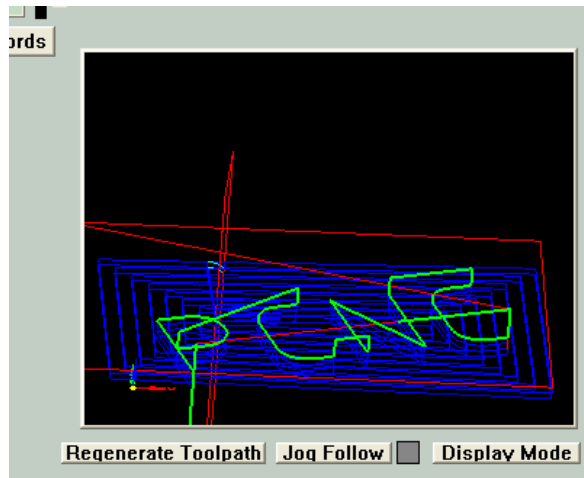


Figure 3.7 – Toolpath during a cut
Using Tormach PCNC 1100

coordinates and your coordinates is called the Work Offset.

Note: We are going to cut this part by giving you a "recipe". You will learn why and how it works later. Please don't add anything extra to the recipe. Adding yeast to a soda bread recipe makes a fine mess. For example, here, you should have the length of each tool in the tool table still set to zero.

You are now ready to run the program.

3.3.3 Cutting in Air

Click the Cycle Start button. The PCNC will start running but almost immediately ask for tool number 1. This is already loaded so just click Cycle Start again. The machine will trace the letters PCNC in the air. It does this four times because they are to be cut 0.2" deep and the program is designed only to cut 0.05" each time. This is conservative but runs no risk of breaking the 1/4" diameter tool.

The toolpath shows the tool as it moves by a different colored line (figure 3.7).

After "cutting" the letters the machine will stop and ask for tool number 2. Jog the Z-axis up a few inches so you can put the 1/2" cutter into the tool holder. Then jog down to about 2" above the material, and type the value 0 into the Z DRO. Do not forget to press *Enter* to accept the value.

Now click Cycle Start again. The machine will use a spiral path to cut out the recess (called a pocket).

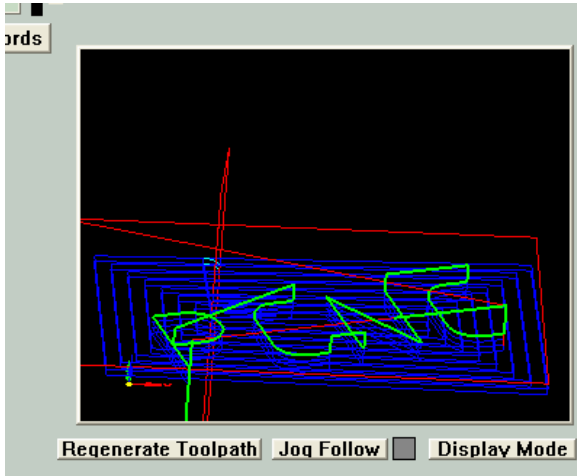


Figure 3.7 – Toolpath during a cut



Figure 3.8 – Touching a tool

3.3.4 Cutting the Actual Part

You should now be ready to actually cut the part.

Jog the Z-axis and swap back to the 1/4" tool. You must now set this so it is just touching the surface of your material and define this as Z = 0. There are many good ways of doing this and we will look at them in a later chapter. For now you just need a thin sheet of paper or perhaps a plastic wrapper from a candy packet.

Jog the Z-axis down to 1/4" or so above the material. If you are using keyboard jogging, change the Slow Jog % DRO from 10% to 2%. The Tormach Jog/Shuttle Controller allows you to control the speed depending on how far you turn the ring.



Figure 3.9 – Cutting the text

Making your first part

Then with the paper below the tool, carefully jog down until the paper just gets trapped by the tool. You must avoid ramming the tool into the material; you want to just touch the paper (figure 3.8).

When in position type 0 and *Enter* into the Z-axis DRO. You now have the Work Offsets set to suit your material thickness and length of tool.

Make certain you have properly shut the door protecting the motor and spindle belts or the spindle will not start and a non-rotating tool will try to cut the work.

Click Cycle Start. The program will stop for a tool change to tool 1. Just click Cycle Start as you have tool 1 already in the spindle.

The PCNC will cut the letters in four passes. Figure 3.9 shows this process in pass two.

When the text is done the machine will stop and request a change to tool 2. Jog up so you can change the tool to the 1/2" cutter. If you hold the *Shift* key down when jogging the jog will take place at full speed rather than the 2% used for very careful positioning. Change the tool and jog down with the paper under the tool. Type 0 and *Enter* into the Z-axis DRO to define the new Work Offsets. Again check that the door is shut.

Click Cycle Start to run the program to cut the pocket. Figure 3.10 shows this partially done.

When the pocket is completed the machine will stop and rewind the program so it can be used to cut another copy of the part.

The completed piece is shown in Figure 3.11. You can compare this with the CAD/CAM visualization in figure 3.7.

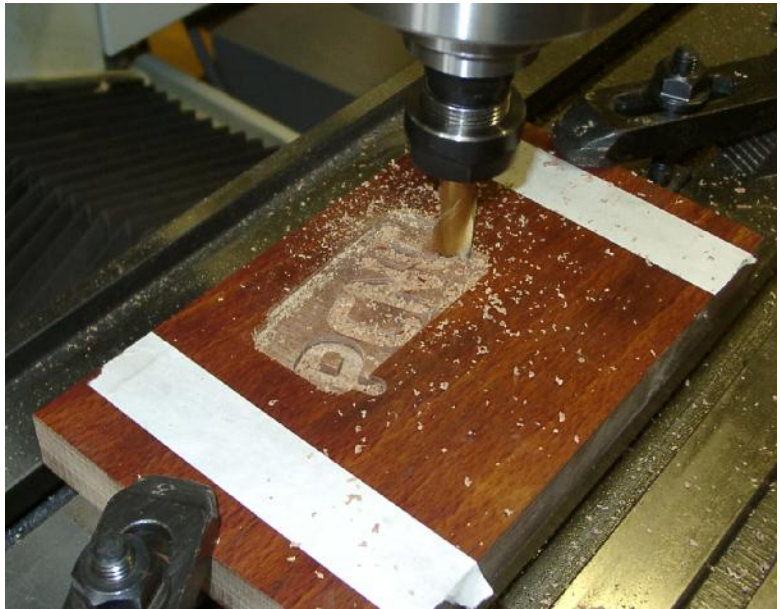


Figure 3.10 – Cutting the pocket

3.3.5 Summary

This chapter has allowed you to make your first part with the PCNC. Although it is fairly simple it would be very difficult to make without a CNC machine. When you are experienced you should be able to design and make something like it from scratch in about 45



Figure 3.11 – The completed part

Making your first part

minutes.

You might have wondered about some of the things that you were told to do. Why did you cut the text then remove half its depth with the pocket? Could the system be told how long each tool is so you do not need to touch tools each time you make a part? Could the part be made in steel with the same program?

These are all good questions. What you were told to do was to make this first part in the easiest way not necessarily the quickest or most accurate. Subsequent chapters will tell you the answers to these questions and give you the understanding to make the right design and manufacturing decisions for yourself. This experience and understanding will be invaluable whether you make production parts yourself or your designs are manufactured by others.

4. Routes from an Idea to a Part

This chapter describes the different ways in which you can define parts to be made with the PCNC 1100.

It explains, in detail, how to use those features which are supplied built-in to or bundled with the PCNC Control Software.

4.1 Controlling the PCNC

If you have run the sample program used in the previous chapter then you will have seen the control language of the PCNC scrolling through the G-code window as it is executed. Although this may look complex it is actually very simple. It was originally designed for machines whose computer was comparable in power to the one in a microwave oven or multifunction TV remote control!

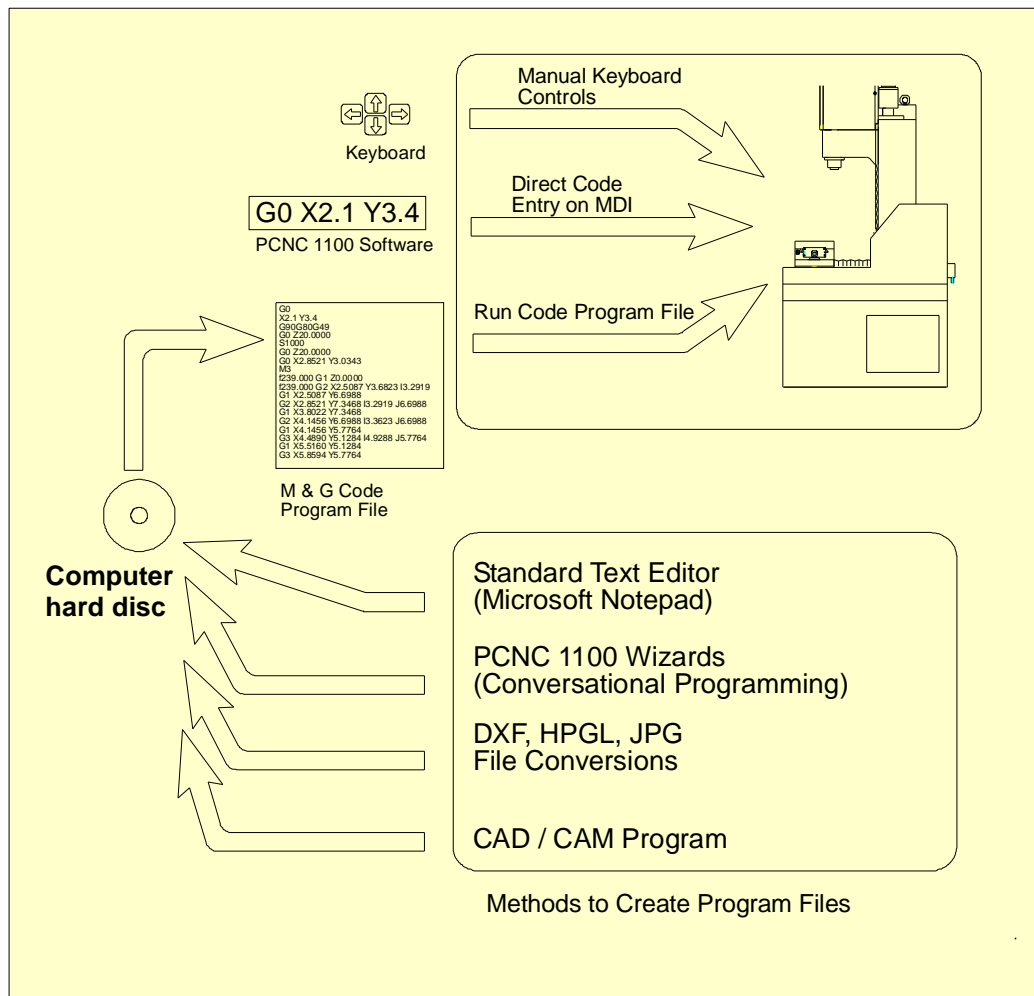


Figure 4.1 – Schematic on how to generate part-programs

The PCNC is always controlled by commands in a part-program. Most of these are G-codes although many other letters are used to define the details like the spindle speed, feed rate, etc. which are required. Thus a part-program is sometimes referred to as a G-code program.

Movement of the machine axes is controlled by three different G-codes.

- G00 moves at the maximum speed of the machine so is used when no cutting is required
- G01 moves in a straight line between two points at the set feed rate
- G02 (and G03) move in a circular arc between two points in a plane and possibly in a straight line in other direction (e.g., to make a spiral).

All the moves take place from the current position of the tool. G00 and G01 need the coordinates of the ending position (e.g., G01 X3.2 Y2.1 Z -0.3 would move the three linear axes together so the tool will follow a straight line ending at X = 3.2, Y = 2.1 and Z = -0.3). G02 and G03, clockwise and counterclockwise arcs respectively, need additional information to define the center point to be used.

Most G-codes are what is termed Modal. This means they stay in effect until another one is used. So if G01 has been used on one line of code then the following lines only need to give the position of the end of the move. As an example the following code will move the tool to a start position (X = 1, Y = 2) and then move it in a square of side 3.1.

```
G00 X1.0 Y2.0
G01 X4.1
Y5.1
X1
Y2
```

In practice, however, the simplicity of G-codes is deceptive. Calculating the position of the “center” of the tool by hand is tedious and error prone. If you wanted to cut around the outside of a block of material to make the 3.1" square then you need to allow for the diameter of the tool and need to move the Z-axis to take acceptable sized cuts in each of several passes.

You will easily be able to learn what a part-program is intended to do but it is unusual to write one by hand as there is a wide range of software tools that can help you (figure 4.1).

4.2 Choosing the Appropriate Design Software

A mill is capable of producing three different classes of work which are usually referred to as 3D, 2½D and 2D.

A 3D product has complex and often smooth curves in all three dimensions. A typical example is a mold for die-casting, vacuum forming or injection molding.

A 2½D product has detail at several depths from the surface of the material but each piece of detail has a flat bottom surface. The FirstPart cut in Chapter 3 is an example of a simple 2½D component.

A 2D product has no changes in depth. This might be a mechanism link cut out of a sheet of steel or lettering cut out of vinyl sheet. 2D products can be milled but are usually produced on specialist machines (e.g., a laser table or plasma cutter).

Design and particularly production of a 3D shape is much more time consuming than something which is 2½D. If the pocket in FirstPart had a dished shape with a fillet in the corners then it would have been cut with a ball nosed tool and would have required many passes a few thou apart rather than the passes about 0.3" apart that could be used with the flat end mill. In addition the design process is complicated because the pocket cannot be defined by a single number (i.e., its depth) if it is 3D.

With current CAD software you will probably find it is easiest to design 2½D parts with a 2½D CAD package or with a full 3D one running in 2½D mode.

4.3 Software for CAD/CAM

This section gives some guidance on things to consider when choosing software. It is a very big topic whose surface we try to skim.

The two most important points are that (a) the PCNC, because of its open and standard architecture, will run the part-programs produced by virtually all Computer Aided Design/Computer Aided Manufacture (CAD/CAM) software and (b) that the more comprehensive the software that you use, the greater will be your and the PCNC's, productivity and the better the quality of the parts made. A common mistake is to purchase inadequate design software and then have to discard it and the investment made in learning to use it.

The Control Software itself includes features for defining and cutting simple parts like keyways, rectangular and circular pockets or a plane face by conversational Wizards. You will be able to do useful work with these features but you will get better control and documentation of your parts by using a dedicated CAD/CAM system. Some software integrates the "drawing" (CAD) and G-code production (CAM) functions but it is equally common to use different programs from different suppliers for the two functions.

It is perfectly possible and often efficient to design using a 3D CAD package and machine with a 2½D CAM package.

4.3.1 3D CAD

3D CAD software has the biggest range of purchase cost and the steepest learning curve. If you want to produce true 3D parts like molds then you have no option but to buy the best package you can afford. Alibre produces a good example of a mid-range 3D CAD package and Alibre Design XPress is bundled with the PCNC 1100.

Alibre allows you to design parts by extruding 2D shapes. The extrusions can be from different planes or surfaces of the part as it is built up and can be solid material or holes.

Several parts can be put together into an assembly and conventional orthographic engineering drawings more or less automatically produced from the model.

High-end programs like Solidworks and Pro/ENGINEER also work in the same way and can be used to design for manufacture with the PCNC 1100.

A 3D CAD program will typically allow the export of solid models (IGES or STL format) for input to a 3D CAM program or DXF format files for use in 2½D CAM.

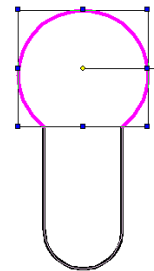


Figure 4.2 – Keyhole

4.3.2 2D CAD

In a 2D CAD program you represent parts by drawing the individual views of the engineering drawings. A vast range of software is available (much of it "free").

We advise that you use software that allows you to express the "intent" of your design in the drawings rather than just drawing a set of lines.

As an example let us assume you want to make a key-hole shaped slot in a block of material. This can easily be drawn as two arcs and two lines with the hole that accepts the key's shank being highlighted (figure 4.2).

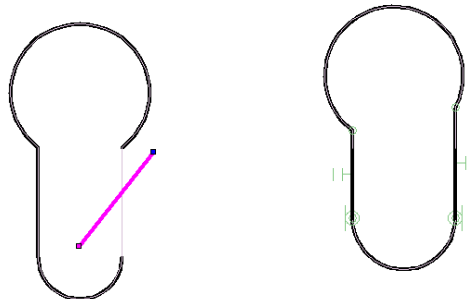


Figure 4.3 – Line moved

Figure 4.4 – Joined points but line moved

How useful this drawing is depends of how intelligently the CAD program can interpret the lines. As drawn they are probably separate and can be moved independently (figure 4.3). Most software will allow joining the lines so they move together (figure 4.4). This however does not really reflect what the part is like.

From an Idea to a part

The important things about the intent of this design are that the straight lines should be parallel and the same length and that they form a tangent to the arc at the bottom of the slot. If these constraints can be given to the CAD software then it will be possible to change its sizes (e.g., the width of the slot by dragging the radius of the bottom arc) while retaining the shape (figure 4.5).

If the shape is defined like this then the software can display the minimum set of dimensions required to define it (figure 4.6). Finally if a dimension is changed then the part can be redrawn to correspond to the new size (figure 4.7).

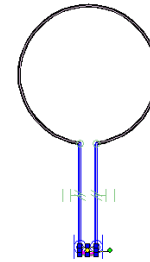


Figure 4.5 – Constrained and size changed

4.3.3 CAM

We have seen the sort of features you can get from a CAD program. There are also big differences between different CAM programs. Some CAM programs accept 2D models (drawings) of your part (e.g., DXF) files and expect you to define the Z depth of features like pickets, engraved text and holes. Alternatively a 3D CAM program will find all the dimensional information in the model

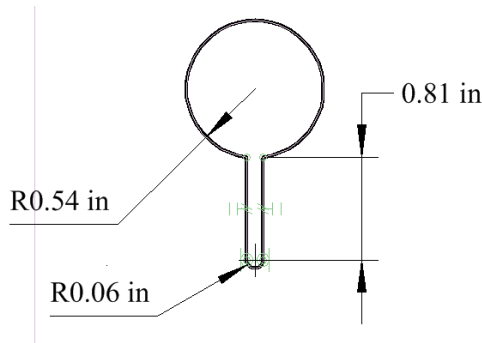


Figure 4.6 – Automatic dimensions displayed

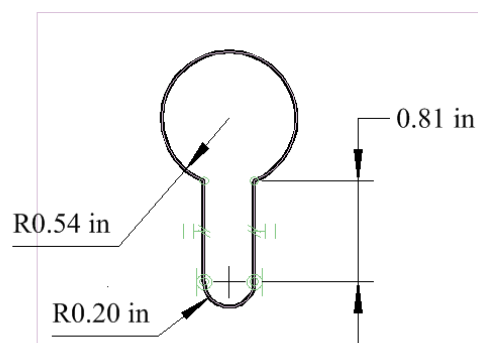


Figure 4.7 – Changed dimension changes drawing

(e.g., IGES) file.

The main issues to consider are how many times you need to tell the CAM program how to machine the part and how good a visualization of the part you will be shown.

Suppose we decide that the inside of the keyhole slot is to be cut with a 0.125" end mill using

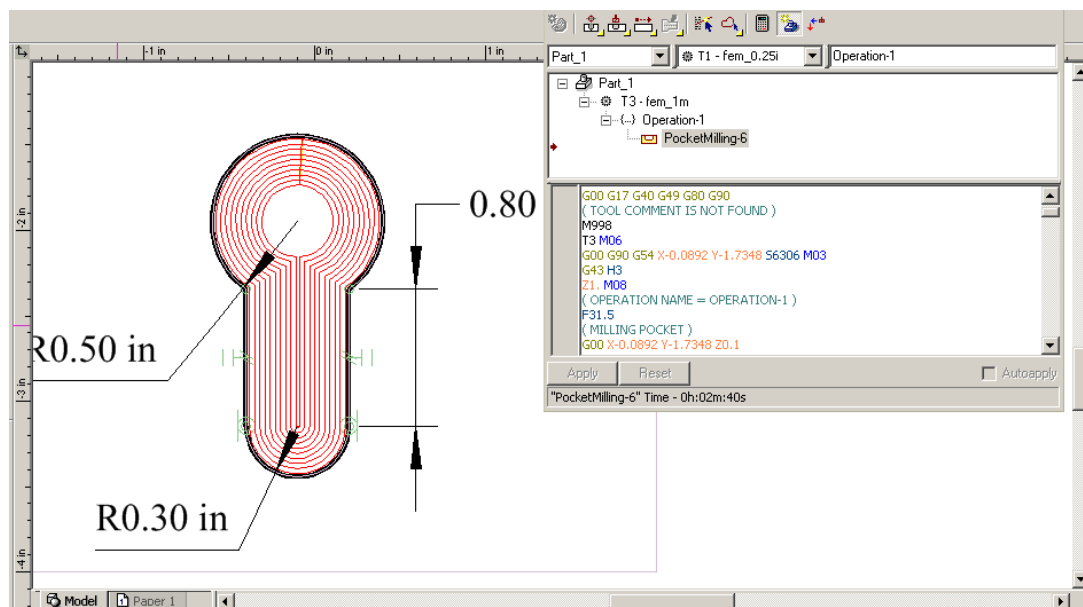


Figure 4.8 – Path of tool and G-code

From an idea to a part

conventional milling to a depth of 0.5" with each pass being 0.1" deep. The CAM software will generate a G-code program for this.

In some freestanding CAM software, if the size of the slot is changed so the drawing changes, then you would have to create the pocket G-code for the slot from scratch with each revision of the drawing. Integrated CAD/CAM software will work dynamically with the CAD and automatically recreate the new G-code with each change in dimension.

A reasonably sophisticated CAM program will give you a choice of views of the finished part and the path of the tool as it is manufactured.

Figure 4.8 is a screen shot of TurboCAD/CAM and shows the toolpath for one version of the slot. It is shown in a plan view together with a pane which displays the G-code generated to perform the operation.

Figure 4.9 is the result of retyping the radius of the end of the slot as 0.15" and displaying a rendered isometric view of the part. The G-code (moved into a smaller pane for clarity on the rendered view) was automatically updated to reflect the new slot width.

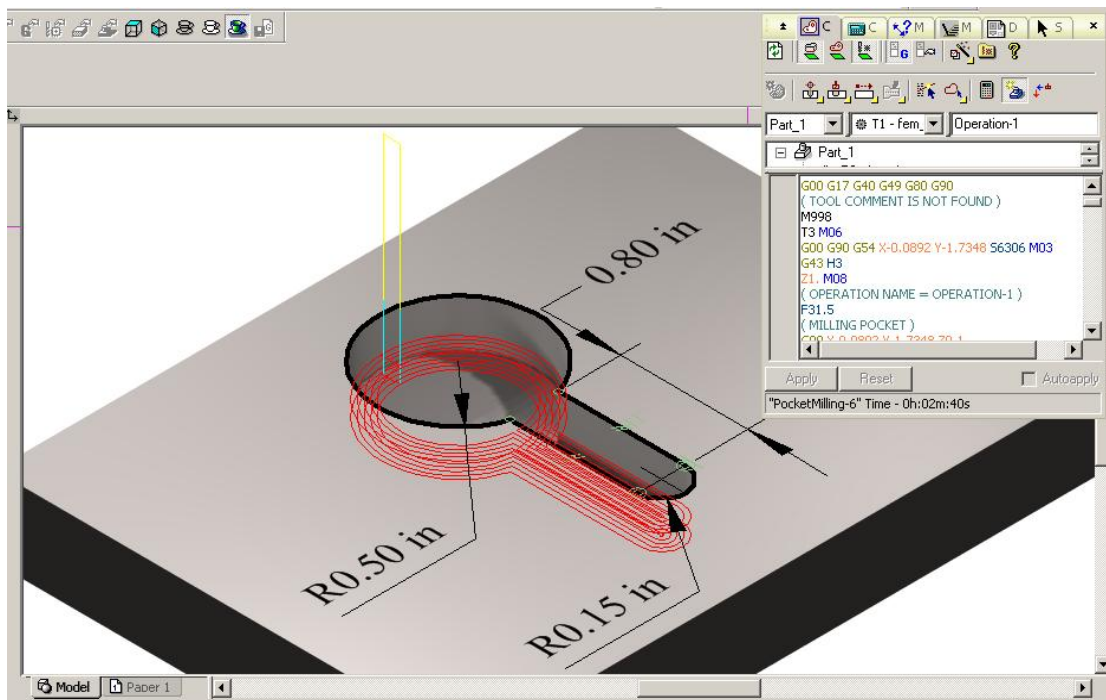


Figure 4.9 – Rendered Isometric view of narrower keyhole slot

4.3.4 Running the G-code

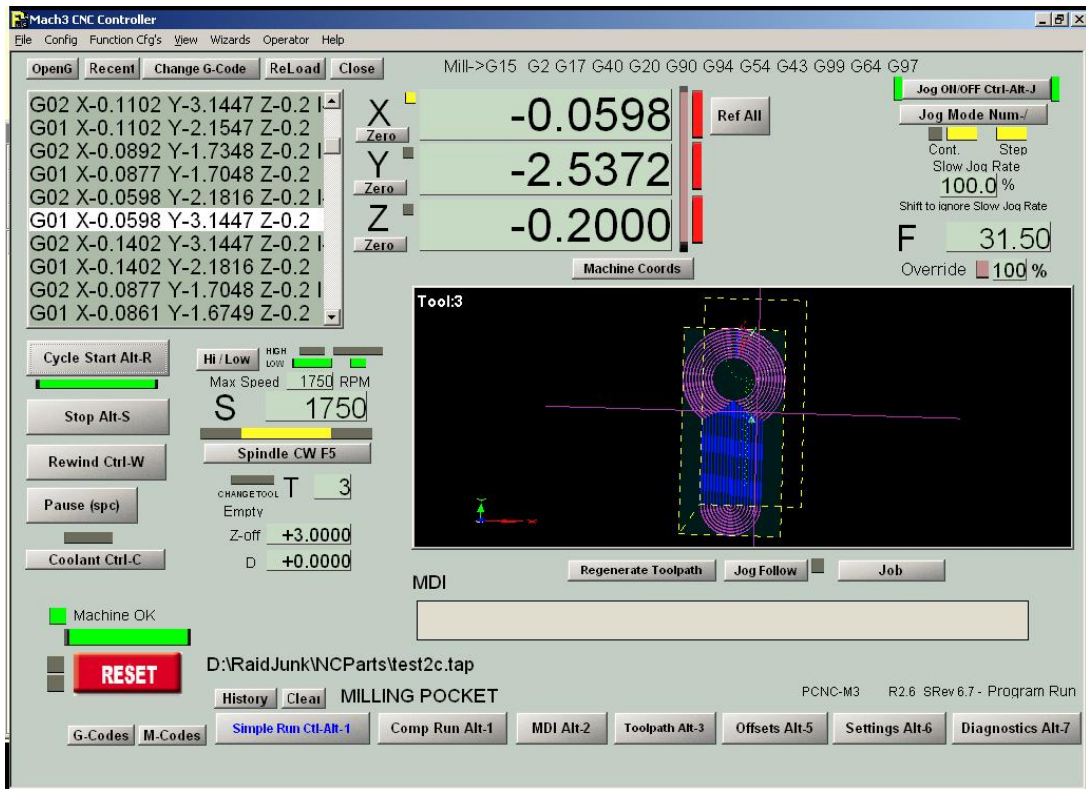


Figure 4.10 – Keyhole loaded and ready to run

The CAM program will write the G-code part-program to disk. Assuming you are using a different computer to design your part from that controlling the machine you need to transfer the file to the hard drive of the control computer. Then you can load and run it in the same way you did for the FirstPart program. Figure 4.10 shows the keyhole loaded into the Control Software.

Note: Do not attempt to run part-programs off a USB key drive or floppy disc.

4.3.5 CAD/CAM Systems

SheetCAM is a 2½D system which will accept DXF drawings from virtually any source, has powerful features and is very easy to learn. An evaluation copy is bundled with the PCNC 1100. If you already have a favorite CAD program then this will be a good choice for you.

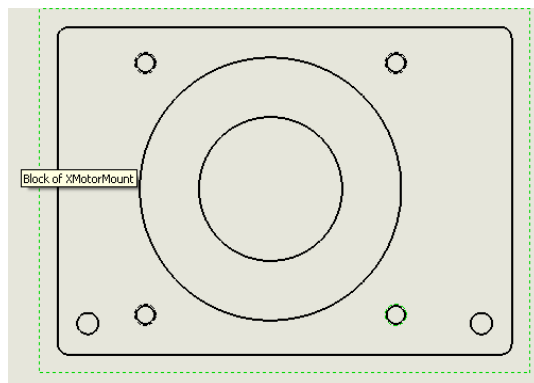


Figure 4.11 – Motor Mount DXF

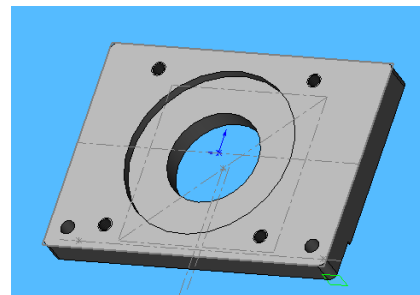


Figure 4.12 – Model for mount

Figure 4.11 shows a DXF drawing of a mounting plate for a motor. This was produced from the solid model in figure 4.12 but could have been drawn with virtually any program.

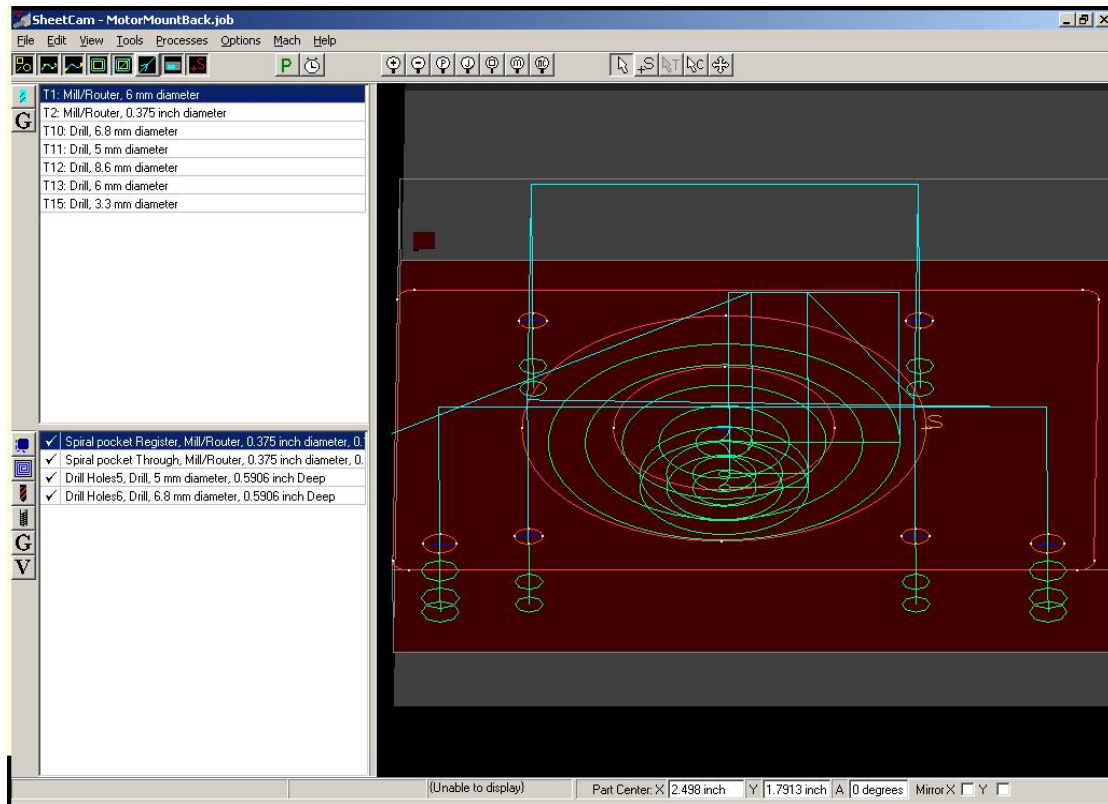


Figure 4.13 – SheetCAM job for Motor Mount

Figure 4.13 shows the drawing loaded as a SheetCAM job

TurboCAD/CAM 3 Professional Edition is an example of a 2½D CAD/CAM system that is very well matched to the PCNC 1100. It offers the dynamic dimensioning and G-code generation described above – indeed the illustrations are screenshots of TurboCAD/CAM. It is a very good value for the cost and Tormach can supply a version specially customized for the PCNC. The illustrations of machining FirstPart show it in use.

If your work needs more powerful parametric solid modeling and 3D milling from these models then the matched combination of Alibre Design and SprutCAM is available from Tormach. Figures 4.14 through 4.16 illustrate a part being designed in Alibre, the manufacture being defined in SprutCAM and a finished test piece.

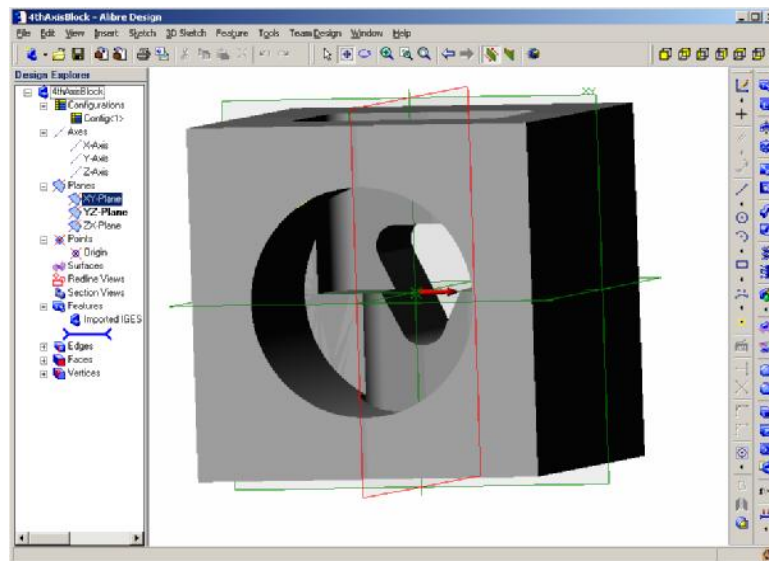


Figure 4.14 – Modelling a 3D part

From an Idea to a part

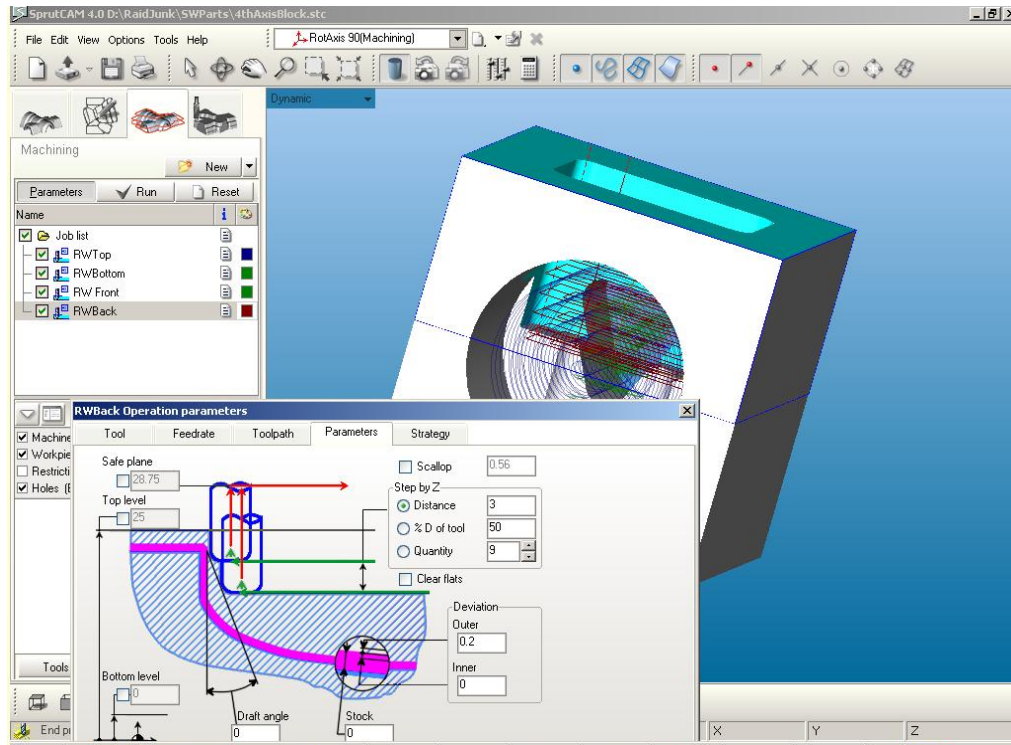


Figure 4.15 – Defining the machining technology



Figure 4.16 – The machined test piece

Virtually any other commercially available CAD/CAM systems can be used with the PCNC control software because of its compliance with industry standards. CAD/CAM software is configured to generate appropriate code for a particular machine tool by a post-processor. You should consult Tormach for advice on the best post-processor option to purchase with your preferred software.

4.4 Programming with Wizards

There will be situations when a simple piece of one-off machining has to be performed and you do not have suitable CAD/CAM software or, perhaps, the machine user is unfamiliar with the software that is available. The Wizard facility built into the Control Software is an ideal solution.

4.4.1 Machining Wizard Concept

A Wizard is a special screen that is displayed on demand by the Control Software. It has a series of DROs with which you define the cuts that need to be made. Figure 4.17 shows the general appearance of a surfacing Wizard.

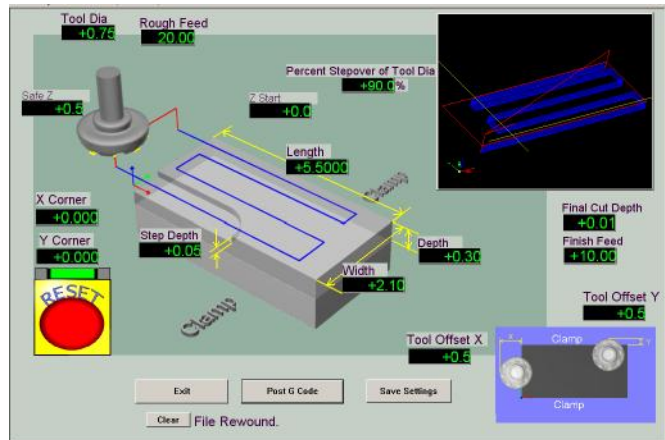


Figure 4.17 – General view of a typical Wizard screen

4.4.1.1 Selecting and Running a Wizard

The Wizards>Pick Wizard menu displays a list of Wizards which are installed on your control computer. Wizards can be obtained from a variety of online sources and you can modify a standard one or write your own from scratch. Figure 4.18 shows an example of what the list could contain.

Click on the Wizard that you want to use and click Run. Notice that a list that is too large to fit in the dialog has a scrollbar but, rather unusually, it is on the left hand side of the list. The Wizard screen will replace the screen currently visible.

4.4.1.2 Standard Wizard Features

Figure 4.19 shows a Wizard for milling a circular pocket.

As many different authors write Wizards, there are detailed differences between the screens but each one should have the following:

- Save Settings button. This causes the Control Software to remember the values in each DRO so they will be available next time you choose the particular Wizard.
- Post Code button. This causes the Wizard to generate a G-code program to cut the shape with the sizes you have specified. Most Wizards have a toolpath window and you will see the cuts to be made in it. You can manipulate the toolpath display in the usual way with the mouse (Rotate, Zoom if Shift depressed and Pan if Right-button drag)
- Exit button. This returns you to the screen that was displayed before you ran the Wizard in preparation for running its code.
- An error line and button to Clear the text from it. This will display any problems found trying to post code using the values that you defined.

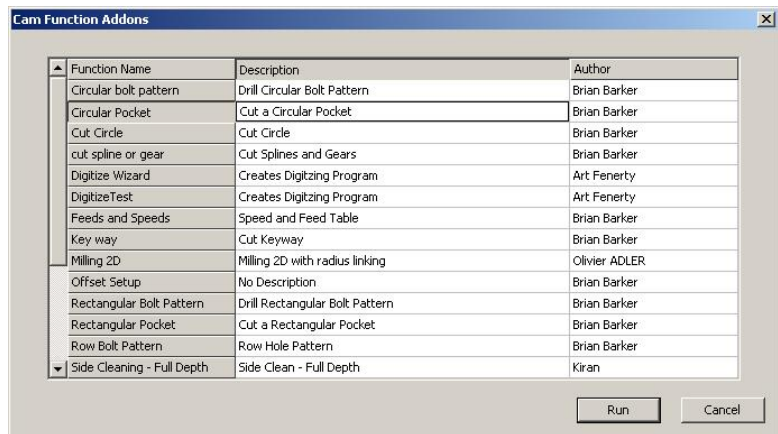


Figure 4.18 – Wizard selection list

Note: A common problem when filling in the DROs of a Wizard is to forget to press *Enter* to accept each value. It is easy to type a value and click in the next DRO. This discards the typed value.

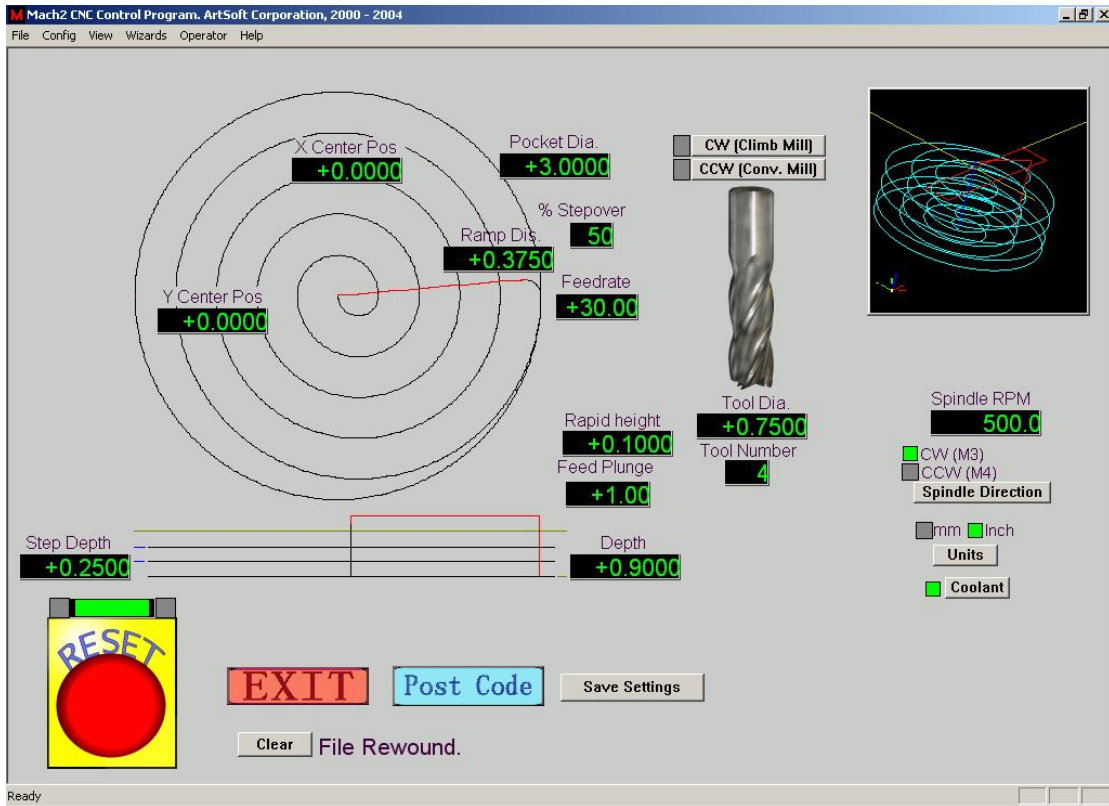


Figure 4.19 – Circular Pocket Wizard

4.4.1.3 G-code from a Wizard

You should carefully check the code generated by a Wizard – particularly if it is one which you have not used before.

Things to look out for are:

- Whether the Wizard sets a speed for the spindle (S-word) and starts it and how it controls the coolant;
- If any motion will conflict with clamps for your work;
- Which position on the work piece the Wizard considers to be $X = 0$, $Y = 0$. A good check is to define $Z = 0$ to be a plane a few inches above your work piece and do a test run of the Wizard's code.

4.4.1.4 Commercial Wizards

While many Wizards are bundled free of charge with the PCNC 1100, a set from New Fangled Solutions is a trial of a product which requires a separate license. You can use all its features to evaluate what the Wizards can cut but, without the license, you cannot actually produce the G-code part-program.

5. Machine Controls

This chapter gives a description of all the PCNC 1100 controls on the operator's panel and the Control Program screens.

If you have been through the section "Running the PCNC" in Chapter 2 you already have a good grounding and may wish to skim through this chapter on first reading of the manual and come back to it when you want to understand exactly how some feature works.

5.1 Machine Operation

5.1.1 Operator's Panel

The operator's panel is illustrated in figure 5.1. It generally controls the PCNC directly and is independent of the software in the control computer. When the **Spindle** and **Coolant** switches are in the **Auto** position then control of these functions is given to the computer.



Figure 5.1 – Operator's panel

Main Switch

The **main switch** is mounted on the right hand side of the control cabinet. A padlock can be inserted to lock it in the off position. It controls the 230 volt power to the main machine and the separate power from the IEC inlet under the control cabinet to the computer and coolant system.

Note: Disconnect the machine from the wall outlets before opening the control cabinet door as it is possible that live terminals can be touched even when the isolator is in the OFF position.

Computer On/Off

This rocker switch controls the power from the IEC inlet to the two outlets that are intended for the computer and monitor. This is provided for convenience. In some applications the computer/monitor might be powered directly from a wall outlet and this "computer" circuit left unused or used for other functions such as to powering a work-light.

Start and EStop

The **start** button will energize the circuits for the axis drives and for the spindle motor. The **stop** button, which locks in once pressed, stops all motion and is the Emergency **stop** (EStop) control. The **stop** button is released by a turning the button-head a quarter-turn clockwise.

Machine start is protected by a no-volt relay from failure of the mains supply. The PCNC requires **start** to be pressed on re-application of power.

Status LEDs

The **Machine** LED indicates that the **start** button has been pressed. When it is lit then the **Machine OK** LED on the computer screen should be solid green.

The **Computer** LED indicates that the Control Software is running.

Spindle

An important safety interlock is that the rotation of the spindle can be disabled (while maintaining axis drive power) by the **Spindle Lockout** key-switch. The switch on the belt-guard door performs the same function. These are used to ensure that the spindle cannot start when an R8 tool holder is being changed (door open) or when a tool is being changed in a collet chuck.

The controls to the right of the key-switch are all concerned with manual control of the spindle. If the **Manual/Auto** switch is in **Auto** position then none of the other panel spindle controls have any effect. In the **Manual** position the computer cannot control the spindle.

The rotary control potentiometer ($RPM \times 100$) controls the speed on the spindle when it is in Manual mode. Its legend indicates approximate spindle speed in the two pulley ranges.

The **Spindle Start** spring loaded rocker starts the spindle and the **Spindle Stop** rocker stops it. The direction is determined by the setting of the **Forward/Reverse** switch. Forward corresponds to clockwise rotation of the spindle. You can safely switch direction while the spindle is turning.

Coolant

The Coolant switch controls the power from the coolant pump outlet on the underside of the control cabinet. In the **Off** position the pump does not run. In the **On** position it runs irrespective of the computer control. In the **Auto** position it will run if the Control Program requests coolant.

Accessory Socket

The Accessory socket is for connection of accessories such as a touch probe.

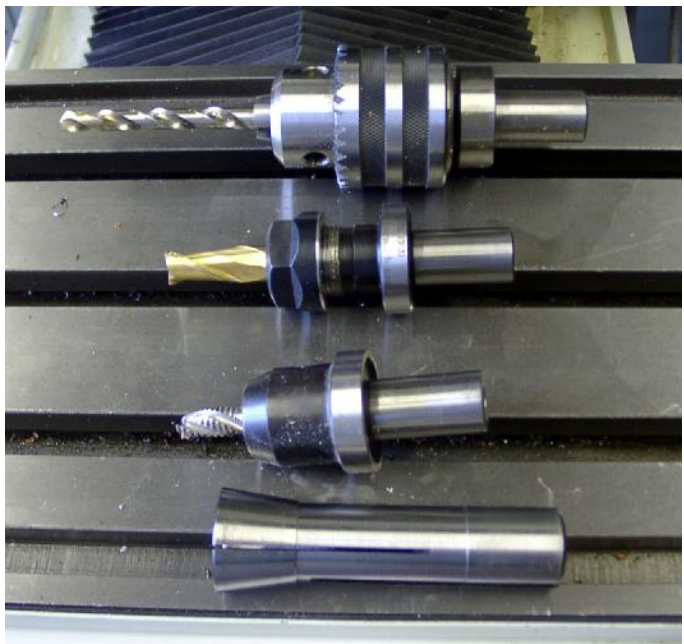


Figure 5.2 – Holders in the TTS range with collet

5.1.2 Tool Changing

5.1.2.1 Tooling Strategy

The PCNC uses a R8 spindle and so you have access to a vast range of standard tooling. The main options open to you are:

- Tormach Tooling System (TTS) – this allows you to have all the tools that you commonly use mounted in low cost tool holders and to change them very quickly. The working length of each tool is accurately maintained each time it is loaded into the spindle so it can be used for cutting with no further setup. Tool lengths can be setup and checked in a stores or inspection area without using the PCNC (figures 5.2 & 5.3).
- R8 tool holders – offer similar features to TTS holders but need to be setup once in the machine and are slower to exchange (figure 5.4).
- Screwed shank cutter collet chuck (Clarkson/Posilok type) – allows you to exchange a cutter while maintaining a known length as the center hole at the back end of the cutter registers in the chuck (figure 5.5).
- ER or similar collet – accepts a wide range of cutter diameters but the length of the tool is not accurately repeatable so the offset needs to be reset after each tool change (figure 5.6).
- R8 collet – gives the shortest overhang from the spindle nose but requires an individual collet for each tool diameter and, as with ER collets, length is not repeatable.

The tool changing procedure is slightly different depending on the tool-holders you have.

5.1.2.2 Changing R8 Tools

Open the spindle drive cover.

Swing the spindle locking fork so it engages with the flats on the upper part of the spindle. You will have to turn the spindle a little to get it to line-up.

The locked spindle is shown in figure 5.7

To remove any installed tools:



Figure 5.3 – TTS presetting



Figure 5.4 – R8 tool-holder

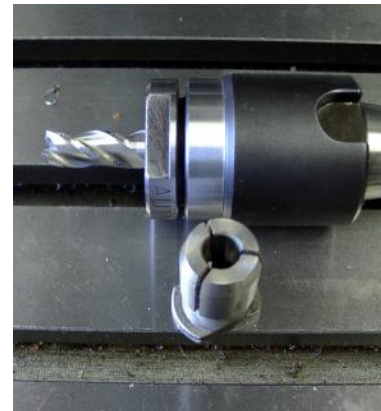


Figure 5.5 – Autolock chuck



Figure 5.6 – ER32 collet tool holder

- Wipe any debris from the spindle nose and the tool being removed to avoid any risk of it getting into the spindle bore;
- Using the supplied 13 mm wrench on the squared end, loosen the drawbar by about one turn. Tap the end of it with a copper-faced hammer to disengage the R8 taper;
- While holding the tooling in one hand, fully unscrew the drawbar and remove the tooling.

To insert a new tool:

- Check that the R8 taper on the tool-holder/collet is quite clean;
- Insert the tool-holder/collet into the spindle turning it to ensure that the drive keyway engages with the key in the spindle. Start the drawbar thread to retain the tool-holder/collet;
- If you are using a tool-holder then the drawbar can be tightened. Otherwise, using the drawbar to prevent the collet jaws from closing in the spindle taper, insert the TTS holder or tool into the collet. Then tighten the drawbar.
- Finally disengage the locking forks and close the door.



Figure 5.7 – Spindle lock fork engaged

5.1.2.3 Changing TTS Tools

Full details of the TTS system are given in its documentation. You should consult the TTS documentation for more information, especially if you are going to make very heavy cuts.

The system uses a precision $\frac{3}{4}$ " R8 collet in the spindle. Insert this as described above for R8 tooling.

Tool holders can be changed by loosening the drawbar by just one turn and using the copper-faced hammer on the squared end to disengage the collet taper. You do not need to remove the collet.

Note: You can change the tools in TTS drill chucks or the ER collet adaptor while these are fitted to the PCNC spindle although doing this will not exploit the pre-settable nature of the tools. You should use both spanners on the ER chuck rather than relying on the holder not turning in the $\frac{3}{4}$ " collet.

If you change a cutter or drill without opening the door to lock the spindle you should use the Spindle Lockout key to avoid any risk of the spindle starting. If the door is open the spindle is automatically inhibited.



Figure 5.8 – Changing spindle speed range

5.1.3 Spindle Speed Ranges

The PCNC has two speed ranges. The low range, 350 (300) to 1450 RPM, is suitable for most machining operations with ferrous and other tough materials. The high range, 900 (800) to 4500 RPM is suitable for small diameter cutters and plastics and non-ferrous materials. The lower of the quoted speeds (in parentheses) is only obtainable when the spindle is under manual control.

The range change is performed by moving the V-belt from the motor from the top pair of pulleys (high speed range) to the lower pair (low speed range).

To change the belt position:

- Open the protective door. Use the rear handle to unlock the motor mounting plate and pull the motor forwards. The belt will slacken and can be moved from one pulley to another (figure 5.8). It is slightly fiddly to get the belt through the slot between the bottom pulley and the head casting.
- Retighten the belt so there is between $\frac{1}{8}$ " and $\frac{1}{4}$ " belt movement midway between the pulleys. Lock the motor mounting and stow the handles in the vertical position.

5.2 Screen Control Panels

5.2.1 Using the Screens

Although at first sight you may feel daunted by the range of options and data displayed by Control Software, this is actually organized into a few logical groups. We refer to these as Families of Controls. By way of explanation of the term “control,” this covers both buttons and their associated keyboard shortcuts used to operate the software **and** the information displayed by DROs (digital read-outs), labels or LEDs (light emitting diodes).

The elements of each control family are defined for reference in this chapter. The families are explained in order of importance for most users.

You should, however, note that all the screens of your **system do not include all the controls** of a family. This may be to increase readability of a particular screen or to avoid accidental changes to the part being machined in a production environment

5.2.2 Families of Related Controls

5.2.2.1 Screen Switching Controls

These controls appear on each screen. They allow switching between screens and also display information about the current state of the system (figure 5.9).

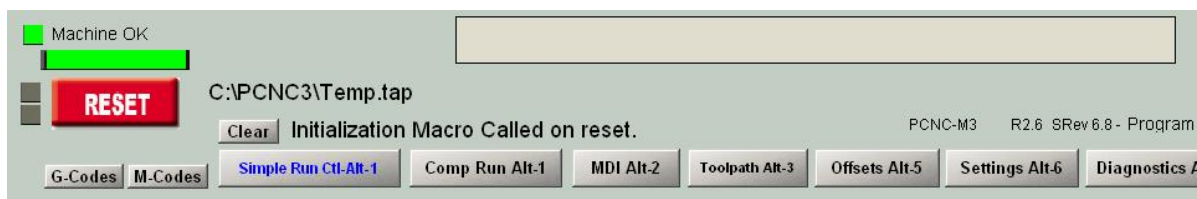


Figure 5.9 – Screen switching control family

Reset

This is a toggle. When the system is Reset the LED glows steadily.

Error and Profile Information

The Machine OK LED indicates that the PCNC is started.

The “intelligent labels” display the last “error” message, the current modes, the file name of the currently loaded part-program (if any) and the Profile that is in use.

The Clear button clears the text from the error line. The entries written to the error line are logged in the file C:\PCNC3\LastErrors.txt. This can be opened and inspected with a text editor like Notepad if you want to analyze a history of events.

Screen Selection Buttons

These buttons switch the display from screen to screen. You are viewing the screen whose label is blue. The keyboard shortcuts are given after the names. For clarity they are letters are shown in upper-case. You should not, however, use the shift key when pressing the shortcut.

5.2.2.2 Axis Control Family

This family is concerned with the current position of the tool, or more precisely, the controlled point (figure 5.10).

The axes have the following controls:

Coordinate Value DROs

These are displayed in the current units (inch/metric = G20/G21). The value is the coordinate of the controlled point in the displayed coordinate system. This will generally be the coordinate system of the current Work Offset (initially 1 – i.e., G54) together with any Tool length offset and G52/G92 offsets applied. The DROs can however be switched to display Absolute Machine Coordinates.

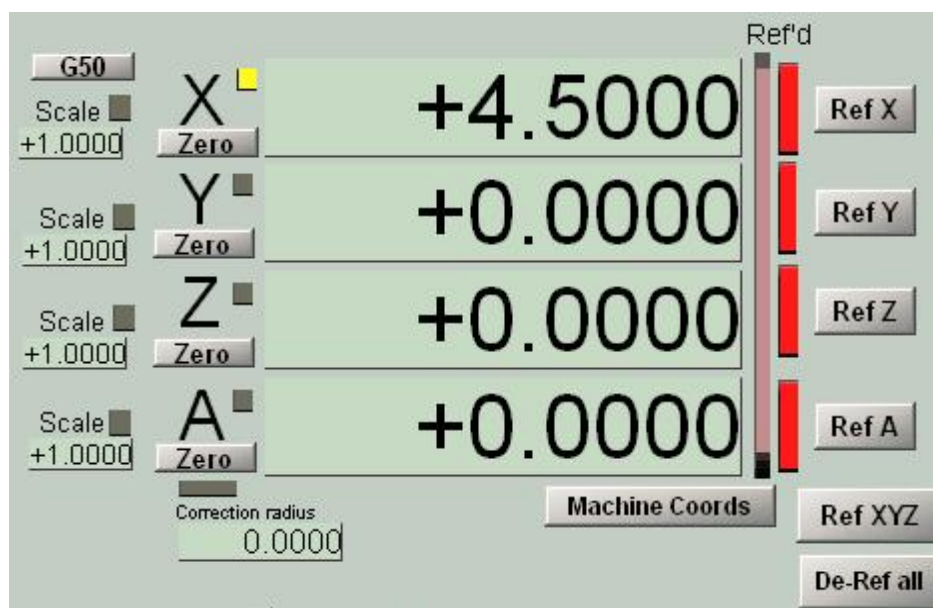


Figure 5.10 – Axis control family

If the X and Y coordinates display in red then they have been rotated by a G68 command.

You can type a new value into an Axis DRO. This will modify the current Work Offset to make the controlled point in the current coordinate system be the value you have set. You are advised to set-up Work Offsets using the Offsets screen until you are fully familiar with working with multiple coordinate systems.

Referenced

The LED is green if the axis has been referenced (i.e., is in a known actual position).

You should **always** run the PCNC in the referenced state.

Machine controls

Each axis can be referenced using its Ref button or the linear axes can be referenced together using the Ref XYZ button.

The De-Ref All button does not move the axes but stops them being in the referenced state.

Machine Coordinates

The Machine Coords button displays absolute machine coordinates. The tall LED warns that absolute coordinates are being displayed.

Scale

Scale factors for any axes can be set by G51 and can be cleared by G50. If a scale factor (other than 1.0) is set then it is applied to coordinates when they appear in G-code (e.g., as X words, Y words etc.) . The Scale LED will flash as a reminder that a scale is set for an axis. The value defined by G51 will appear and can be set, in the Scale DRO. Negative values mirror the coordinates about the relevant axis.

The G50 button executes a G50 command to set all scales to 1.0

Correction Radius

Rotary axes can have the approximate size of the work piece defined using the Rotational Diameter control family. This size is used when making blended feed rate calculations for coordinated motion including the 4th Axis. The LED indicates that a non-zero value is defined.

Jog/Shuttle Axis

The axis that is selected for jogging by the Tormach Jog/Shuttle Controller is indicated by a LED by the top left corner of the corresponding DRO.

5.2.2.3 Jogging Control Family

Jogging can be performed in two ways: (a) using the keyboard (or optional Keypad pendant) or (b) Using the optional Tormach Jog/Shuttle Controller

Keyboard Jogging

Whenever the Jog ON/OFF button is displayed on the current screen then the axes of the machine can be jogged using the jog hotkeys on main keyboard or a pendant keyboard

If the Jog ON/OFF button is not displayed or it is toggled to OFF then keyboard jogging is not allowed for safety reasons.

There are two modes, Continuous and Step which are selected by the Jog Mode button and indicated by the LEDs.

Continuous mode moves the axis or axes at the defined slow jog rate while the hotkeys are pressed.

The continuous jog speed is defined as shown below but this can be overridden by pressing *Shift* with the hotkey(s). A LED beside the Cont. LED indicates this full speed jogging is selected.

The jogging speed used with hotkeys in Continuous mode is set as a percentage of the rapid traverse rate for the axis and for the Windows compatible joystick as a percentage of the feed for the given stick deflection by the Slow Jog Percentage DRO. This can be set (in the range 0.1% to 100%) by typing into the DRO. It can be nudged in 5% increments by the buttons or their hotkeys.

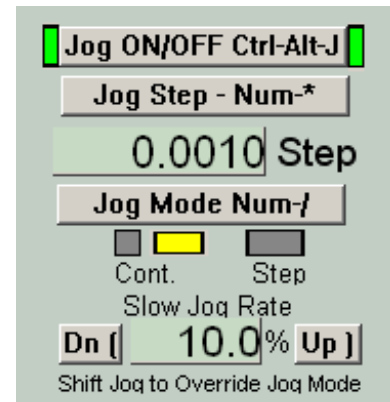


Figure 5.11 – Jogging control family

Step mode moves the axis by one increment (as defined by the Step DRO) for each key press. The **current** feed rate (as defined by the F word) is used for these moves. The size of increment can be set by typing it into the Step DRO or values can be set in this DRO by cycling through a set of predefined values using the Jog Step button.

Incremental mode is selected by the toggle button or, if in Continuous Mode temporarily selected by holding down *Ctrl* before performing the jog.

Tormach Jog/Shuttle Controller

This jogging device is available as an optional accessory (part number 30616, figure 5.12).

Many users will find that it increases their productivity, especially on short-run jobs requiring a lot of setting up of workpiece and tooling.

The four buttons starting at the left are allocated to select jogging of axes X, Y, Z and A respectively. A LED beside an axis DRO indicates that this axis is the one that will be jogged.

The fifth button cycle through the available jog step sizes. Alternatively you can type a step size into the Step DRO.

Continuous jogging is performed by turning the spring loaded ring counterclockwise for the minus direction and clockwise for the plus direction. There are seven speeds, in geometric ratio, from very slow to full speed so you can position any axis with great speed and precision.

The inner wheel (with finger dimple) will jog by one Step as defined in the Step DRO for each click. The move will be made at the current feed rate.



Figure 5.12 – Tormach Jog/Shuttle Controller

5.2.2.4 Spindle Speed Control Family

The machine spindle can be controlled in two ways: (a) by hand or (b) by switching to Auto on Operator's panel, thus making the speed and direction set by the Control Software.

This control family is only important for case (b).

The S DRO has its value set when an S word is used in a part-program. It is the desired spindle speed. It can also be set by typing into the DRO (figure 5.13).

It is an error to try to set it (in either way) to a speed greater than that displayed in Max Speed for the chosen pulley and the nearest legal value will be chosen.

The Hi/Lo button toggles between the two belt/pulley settings with the LEDs indicating the configuration (spindle on left, motor on right)

The maximum available speed is indicated.

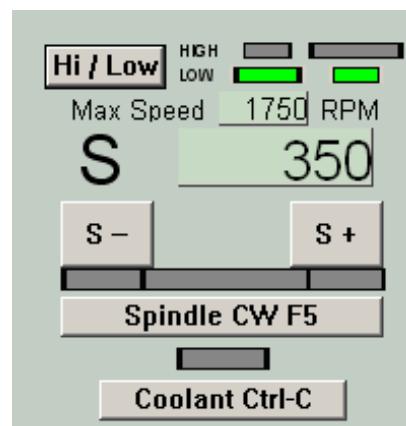


Figure 5.13 – Spindle speed control family

Machine controls

The S- and S+ buttons increase and decrease the value in the S DRO geometrically using a ratio of 1.15 for each click on the button. This reflects the usual arrangement of gearbox in a manual machine tool.

The spindle can be started in a clockwise direction by the Spindle CW button. An M-code (M4) in the MDI box is required if you want to select counterclockwise running.

The main LED above the Spindle CW button indicates that the spindle is “running” – although actual movement may be inhibited by the key and/or door interlocks. The two LEDs that flank it indicate a Dwell while the spindle gets up to speed or fully stops.

The Coolant button and its associated LED control the coolant power outlet under the control cabinet.

5.2.2.5 Feed Control Family

Feed Units per Minute

The F DRO gives the feed rate in current units (inches/millimeters per minute). It is set by the F word in a part-program or by typing into the F DRO. The control software will aim to use this speed as the actual rate of the coordinated movement of the tool through the material. If this rate is not possible because of the maximum permitted speed of any axis then the actual feed rate will be the highest achievable (figure 5.14).

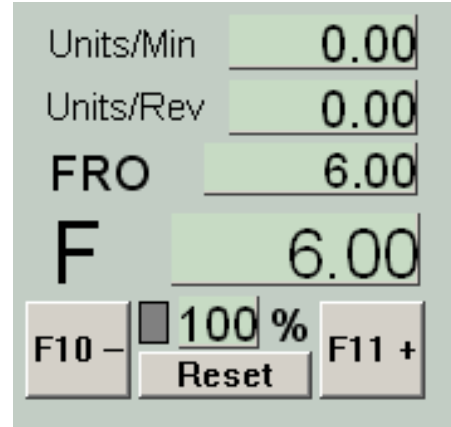


Figure 5.14 – Feed control family

Feed Units per Revolution

As modern cutters are often specified by the permitted cut per “tip” it may be convenient to specify the feed per revolution (i.e., feed per tip x number of tips on tool). In this case the F DRO gives the feed rate in current units (inches/millimeters) per rev of the spindle. It is set by the F word in a part-program or by typing into the DRO.

Notice that the numeric values in the control will be very different unless spindle speed is near to 1 rpm! So using a feed per minute figure with feed per rev mode will probably produce a disastrous crash!

Feed Display

The actual feed in operation allowing for the coordinated motion of all axes is displayed in Units/min and Units/rev. If the spindle speed is not set and the actual spindle speed is not measured then the Feed per rev value will be meaningless.

Feed Override

Unless M49 (disable feed rate override) is in use, the feed rate can be manually overridden in the range 20% to 299% by entering a percentage in the DRO. This value can be nudged (in steps of 10%) with the buttons or their keyboard shortcuts and be reset to 100%. The LED warns if an override is in operation.

The control software will apply any changes in feed rate override as quickly as possible. This could be very important if you decide you have too high a feed rate for safety. To optimize performance, the software keeps a queue of moves ready to be implemented. It is possible if you increase the feed rate that you will make one of these queued moves be faster than the PCNC can move or accelerate and so it will lose steps. The best strategy is to write the part program to run at the highest anticipated feed rate and use the override to reduce it produce the best operating conditions

The FRO DRO displays the calculated result of applying the percentage override to the set feed rate.

5.2.2.6 Program Running Control Family

These controls handle the execution of a loaded part-program or the commands on an MDI line (figure 5.15).

Cycle Start

Safety warning: Note that the Cycle Start button will, in general, start the spindle and axis movement.



Figure 5.15 – Program running family

Stop

Stop halts axis motion as quickly as possible. Unless used when Paused, it may result in lost steps (especially on stepper motor driven axes) and restarting may not be valid.

Rewind

Rewinds the currently loaded part-program.

Single BLK

Single BLK is a toggle (with indicator LED). In Single Block mode a Cycle Start will execute the next single line of the part-program.

Pause

Pause brings the current move to a controlled stop applying deceleration etc. Resumption by Cycle Start is always safe. Although jogging while paused is possible we do not recommend doing it.

Line Number

Line DRO is the ordinal number of the current line in the G-code display window (starting from 0). Note that this is not related to the “N word” line number.

You can type into this DRO to set the current line.

Run from Here

Run from here performs a dummy run of the part-program to establish what the modal state (G20/G21, G90/G91, etc.) should be and then prompts for a move to put the controlled point in the correct position to for the start of the line in Line Number. You should not attempt to Run from here in the middle of a subroutine.

Set next line

Like Run from here but without the preparatory mode setting up processing.

Block Delete

The Ignore "/" Blocks button toggles the Block Delete “switch.” If enabled then lines of G-code which start with a slash (i.e., / - will not be executed).

Optional Stop

The M01 Break button toggles the Optional Stop “switch.” If enabled then the M01 command will be treated as M00.

Goto Toolchange

This button provides manual movement of the controlled point when the part-program is stopped.

Tool Details

Controls display the current tool, its name, the offsets for its length and diameter and, on systems with a Digitizer input, allow it to be automatically zero to the Z plane.

Unless tool change requests are being ignored, on encountering an M6 the Control Software will stop and flash the Change Req LED. You continue (after changing the tool) by clicking Cycle Start.

Work Offset Details

The name of the current work offset coordinate system is displayed.

5.2.2.7 Toolpath Control Family

The currently loaded part-program is displayed in the G-code window. The current line is highlighted and can be moved using the scroll bar on the window (figure 5.16).

The Toolpath display shows the path that the controlled point will follow in the X, Y and Z planes. When a part-program is executing, the path is over painted in green. This over painting is dynamic and is not preserved when you change screens or indeed alter views of the toolpath.

On occasions you will find that the display does not exactly follow the planned path. This occurs because the Control Software prioritizes the tasks it is doing. Sending accurate step pulses to the machine tool is the first priority. Drawing the toolpath is a lower priority. It will draw points on the toolpath display whenever it has spare time and it joins these points by straight lines. So, if time is short, only a few points will be drawn and circles will tend to appear as polygons where the straight sides are very noticeable. This is nothing to worry about.

The Simulate Program Run button will execute the G-code, but without any tool movement and allow the time to make the part to be estimated.

The Absolute Motion Extremes data allow you to check if the maximum excursion of the

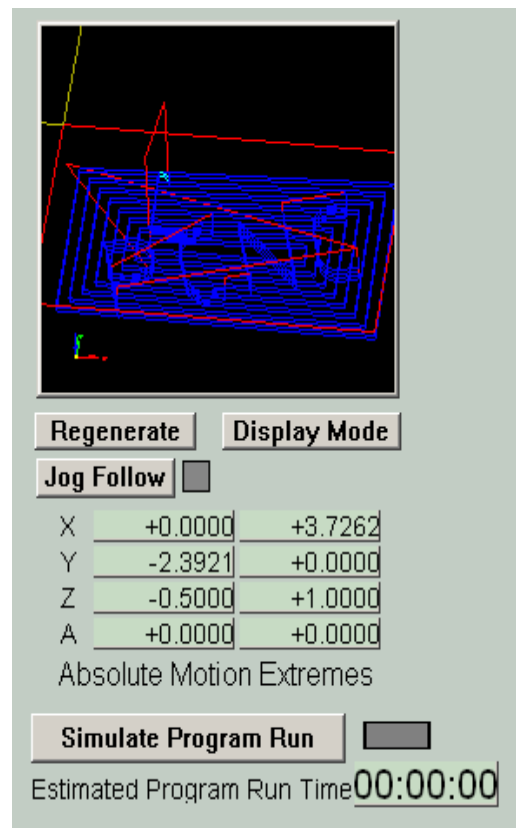


Figure 5.16 – Toolpath and G-code display family

controlled point is reasonable (e.g., not milling the top of the table).

The toolpath display can be rotated by left clicking and dragging the mouse in it. It can be zoomed by shift-left clicking and dragging and can be panned by dragging a right click.

The Regenerate button will regenerate the toolpath display from the G-code with the currently enabled fixture and G92 offsets.

The Display Mode button will chose whether the default toolpath display is sized for the machine envelope (as defined by the soft limits) or the object defined by the extremis of the part-program.

The Jog Follow button allows the toolpath display to be automatically scrolled as the controlled point is jogged.

5.2.2.8 File Control Family

These controls are involved with the file of your part-program. Most should be self-evident in operation.



Figure 5.17 – File control family

Change G-code allows you to use a variety of tools to edit the part-program you have loaded. When clicked it displays a dialog allowing you to choose the program to update the G-code (figure 5.17).

Clicking a button will run the named CAM program (or editor) and load the file corresponding to the G-code into it (figure 5.18). You must, of course, specify the program that created the G-code in the first place. For example, running SprutCAM with code produced by SheetCAM will give a “file not found” error and an untitled CAM job.



Figure 5.18 – Redirector to edit G-code

When you have recreated the G-code in the CAM software and exited (or exited from the editor) then the new file will automatically be loaded into the control

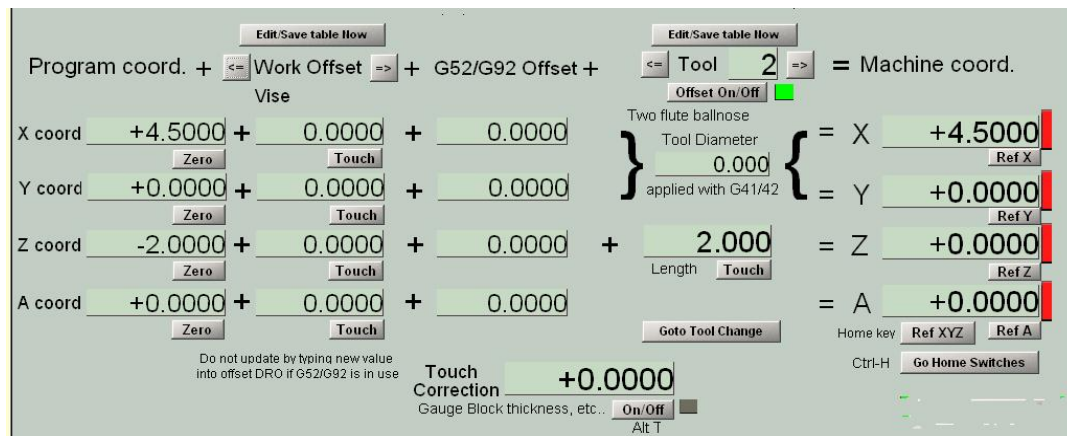


Figure 5.19 – Offsets control family

software.

SheetCAM works in a slightly different way as it will reload the G-code as soon as you run its post processor function and you do not need to close SheetCAM down. This allows very efficient fine tuning of the machining process.

5.2.2.9 Work Offset and Tool Table Control Family

This family is explained in detail in Chapter 6 (figure 5.19).

5.2.2.10 MDI and Teach Control Family

G-code lines (blocks) can be entered, for immediate execution, into the MDI (Manual Data Input) line. This is selected by clicking in it or the MDI hotkey (*Enter*). When the MDI line is active its color changes and a fly-out box showing the recently entered commands is displayed (figure 5.20). The cursor up and down arrow keys can be used to select from the fly-out so that you can reuse a line that you have already entered. The *Enter* key causes the current MDI line to be executed and the MDI remains active for input of another set of commands. The *Esc* key clears the line and de-selects it.

You need to remember that when it is selected all keyboard is written in the MDI line rather than controlling the machine. In particular, jogging keys will not be recognized – you must *Esc* after entering MDI.

The Control Software can remember all the MDI lines as it executes them and store them in a file by using the Teach facility.

Click Start Teach, enter the required commands and then click Stop Teach. The LED blinks to remind you that you are in Teach Mode. The commands are written in the file with the conventional name C:\PCNC3\GCode\MDITeach.tap. Clicking Load/Edit will load this file so it can be run or edited in the usual way – you need to go to the Simple Run or Comp Run screen to see it. If you wish to keep a given set of taught commands then you should Edit the file and use Save As in the editor to give it your own name and put it in a convenient folder.

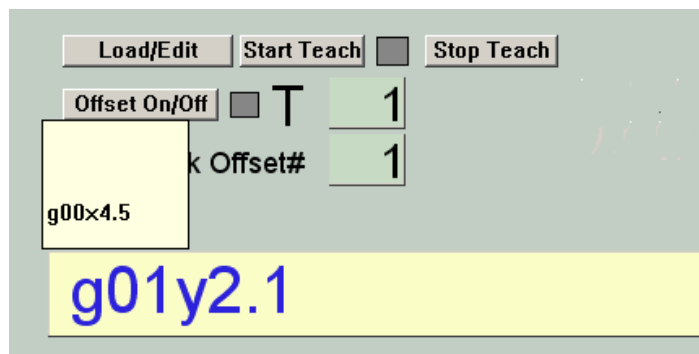


Figure 5.20 – MDI and Teach control family

5.2.2.11 Loop Control Family

The Control Software can execute a part-program many times, automatically updating a limit to the depth to which the Z-axis can move (figure 5.21).

The part-program must end with an M30 (rewind) code.

The number of passes you require is entered into the Cycles DRO. The initial lowest Z position is put into Z inhibit (note this will often be negative) and the distance Z is to move down each cycle is put into Z Step (usually positive). The On/Off button enables the feature.

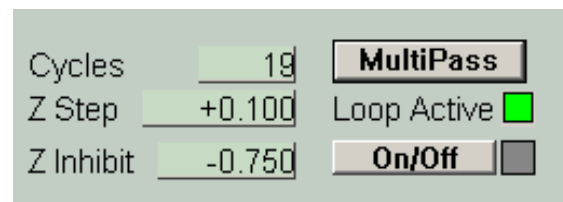


Figure 5.21 – Loop control

The Multipass button will prompt for the values by a series of dialogs rather than requiring the

Mill->G15 G80 G17 G40 G20 G90 G94 G54 G49 G99 G64

Figure 5.22 – Current modes display

DROs to be filled in.

5.2.2.12 Modes and Mode Alarm Control Family

These families display the current modes of the Control Software and an “alarm” LED for unusual modes. The alarm is on the Diagnostics screen. Click on the flashing LED to see a list of unusual modes (figure 5.22).

These modes might arise from running a part-program produced by a CAD/CAM system not customized for all the features of the PCNC.

An unusual mode does not imply a fault in your system, simply a hint of what to look for if unusual things are happening (figure 5.23).

If for any reason the standard modes do not suit your normal working then by double-click on the unusual modes LED you can define the current state as “usual.”

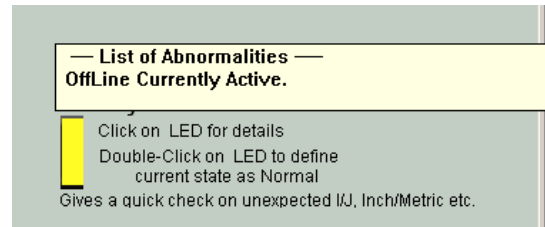


Figure 5.23 – Unusual modes display on Diagnostics

5.2.2.13 Rotational Diameter Control Family

As described in the Feed Rate control family, it is possible to define the approximate size of a rotated work piece so the rotational axis speed can be correctly included in the blended feed rate. The relevant diameters are entered in the DRO on this family (figure 5.24).

The Axis control Family has a warning LED to indicate the setting of a non-zero value here.

Note: A value is not required if rotary movement is not to be coordinated with linear axes. In this case a suitable F word for degrees per minute or degrees per rev should be programmed.

5.2.2.14 Toolchange Position Control Family

This family defines the place the machine should go to when a request to change a tool occurs or when the Goto Toolchange button is clicked. A part-program can go to it by using the M998 code (figure 5.25).

The position is defined in machine coordinates (i.e., relative to the home switches) and in the default machine units.

Enter the special value 9999 if you do not want a given axis to move when a tool change is requested.

The units in which you define your toolchange position can be configured by the Toggle T/C Units button.

Define Toolchange position units (mm/inch) d

Toggle T/C Units Currently set for: inches

Tool Change Location		G28 Home coordinates
Absolute coordinates from Home in: inches		
X	+9999.0000	+0.0000
Y	+9999.0000	+0.0000
Z	-3.0000	+0.0000

9999 = do not move axis

Figure 5.25 – Tool change position

Note: The move will take place correctly whatever units the machine is in at the time and, because machine coordinates are used, to a fixed place irrespective of the offsets in use.

The tool change position is also used when you want to use the central lubrication system.

This family also allows you to set the machine coordinates used if your part-program executes a G28. We advise you to use the M998 toolchange location whenever possible as this checks that the machine is referenced (i.e., the machine coordinates are valid). G28 will use whatever values are in use irrespective of them being valid because the machine is referenced.

5.2.2.15 Inhibits and Overrides Control Family

In testing a part-program it is sometimes useful to be able to inhibit the movement of an axis (e.g., typically Z). This is done with the Axis Inhibit buttons (figure 5.26).

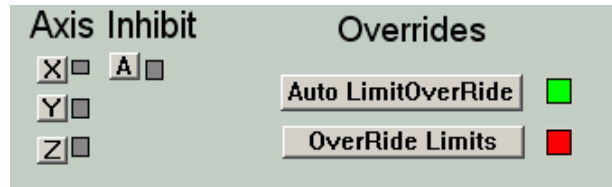


Figure 5.26 – Inhibits and overrides family

When an axis hits a limit switch it will trigger a software **Estop** condition so all movement will cease. If Auto LimitOverride is enabled then you can reset the system and carefully jog off the limit. If you want more security in this situation you can disable AutoLimitOverride and click the OverRide Limits button before clicking Reset on each occasion.

5.2.2.16 Feeds and Speeds Calculator

Different materials and tooling require different cutting speeds and feed so you will need to use published tables and your experience to define the optimum spindle speed and the feed rate for any given job. A simple calculator on the Settings screen will, however, do the math for you when you know a cutting speed (in feet per minute) and a chip per tooth loading (figure 5.27).

Figure 5.27 – Speed/Feed calculator

Enter your cutting data into the DROs, not forgetting to press Enter for each value and click the Calc RPM/Feed button. The screen labels show the calculation that is being done for you.

5.2.2.17 Tapping Configuration Family

The control software allows use of the Tormach Tapping Heads (PN 30612 and 30613) without a detailed understanding of the sequence of moves that are needed to control the head. The holes are tapped by using one of the M-code macros M871, M872, M873 or M874. The macros work as a kind of canned cycle. Each macro is set up for a particular pitch of thread and choice of tapping head. This information is provided by the Tapping Configuration family of controls (figure 5.28).

Figure 5.28 – Tapping Configuration

The TPI or Pitch DRO is interpreted in a different way depending on whether the part program calling its macro is running in Inch (G20) or metric (G21) mode. In G20 mode the value is a number of threads per inch and in G21 mode it is the pitch of the thread in millimeters. It is

perfectly possible to change modes within a program and so have a job with mixed thread standards. Fortunately Inch numbers are unlikely to be pitches and vice-versa. The control software checks that the chosen pitch can be cut at the current spindle speed when the macro is called so there is little chance of damage to a tap.

Both tapping heads have a high-speed reverse feature to minimize tapping time. As the gear ratios are different for each head you must define which is going to be used by toggling to the appropriate large or small LED.

5.2.2.18 Misc. Settings Control Family

Current Units

This displays the units currently selected (figure 5.29).

Ignore Tool Change

Normally when the part-program calls for a different tool execution will stop, the Tool Change LED will flash and you need to press Cycle Start to continue. Tool change requests can be ignored by clicking the Ignore M6 Tool Change button. This can be useful when air-cutting to prove/time a part program.

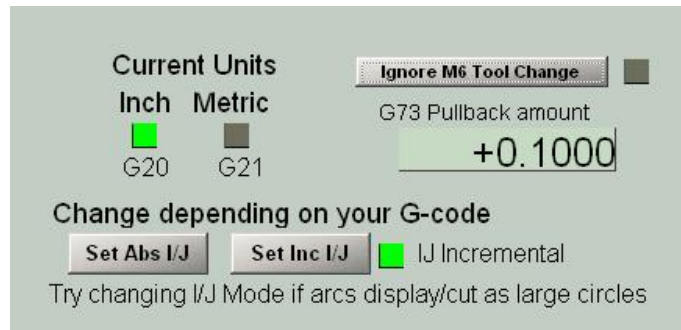


Figure 5.29 – Misc. settings

G73 Pullback

This DRO defines the distance that a high speed peck drill cycle (G73) will pullback when breaking the chip. You may need to alter this value for metric use.

IJ Mode

The IJ mode defines how the I, J and K words in a G02 or G03 are interpreted. There is no standard for this so different CAM post processors will use different conventions. The PCNC 1100 usually runs with I, J and K being **incremental** distances from the current point. You can make the Control Software treat them as absolute positions by clicking the Set Abs I/J button. The symptoms of an incorrect setting are that small arcs can display a massive circle on the toolpath or you get an error message about the radius of an arc being different at the start and end of a cut.

5.3 USB Jogging Pendants

Optional accessories purchased by many users are the USB jogging pendants. These provide access to the main axis jogging controls on a unit that can be positioned for use while touching-off the tool position. They are not resistant to coolant and, for safety reasons, do not have controls for starting and stopping machining.

The pendants can be plugged into any spare USB port. One on the keyboard is often convenient. Windows will recognize it and install any driver software automatically.



Figure 5.30 – Jog Wheel pendant

5.3.1 Jog/Shuttle Controller

The Jog /Shuttle Controller is a low cost but very precise and fast way of jogging the four axes. There are five buttons and two rotary controls (figure 5.30).

Machine controls

The buttons allow you to select which axis you wish to jog. The chosen axis is indicated by a LED next to the corresponding DRO. For example, figure 5.31 shows that the X-axis is the one to be jogged. If a Jog/Shuttle Controller is not installed then all the LEDs will be dark.

The Step button will cycle through a set of jog step sizes in the Step DRO in the Jogging family.

The outer rotary shuttle control is spring-loaded. Turning it will jog the selected axis at a speed proportional to the amount it is turned. There are seven different speeds arranged geometrically so you have instant control from the full rapid speed (65 rpm) down to the slowest crawl.

The inner rotary control, operated by fingertip, has detents. Each click will jog at the current feed rate by the distance defined in the Step DRO. If you turn it faster than the axis can move the impossible steps will be ignored so the axis will never buffer up a backlog.

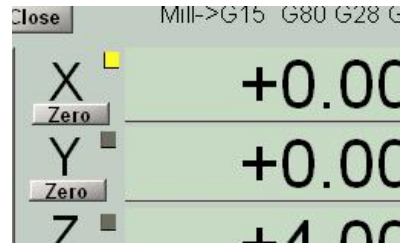


Figure 5.31 – X-axis will be jogged

5.3.2 Keypad Pendant

The keypad pendant has two modes of operation. The CNC/Num key toggles between these modes (figure 5.32).

When the Numeric LED is lit it is a conventional numeric pad. Note that as Backspace is not available on the main keyboard during DRO entry it cannot be used on the pendant either.

When the LED is not lit the majority of the keys are concerned with jogging the four labeled axes (X, Y, Z and A – keys 2, 3, 4, 6, +, 8, 9, -)

The 7-key will perform a Home All operation (homing Z to limit switches and then X and Y).

The 1-key will toggle the state on the Optional Stop LED. When the LED is lit, M01 is treated exactly like M00 (i.e., end program), otherwise M01 is ignored.

The /-key toggles between Continuous and Incremental (or Step) jogging modes. In step, one key press produces a single step axis movement. The step size is displayed in a DRO.

The *-key (labeled Size) cycles through the available Increment (Step) sizes.



Figure 5.32 – Keypad pendant

6. Using Multiple Tools

This chapter explains how you can configure the PCNC 1100 to use several tools without having to waste time at each tool change defining the length of the new tool. It also covers techniques for simplifying machining of work in a vise or other fixture.

Offsets used are often considered as advanced features but the design of the PCNC Control Software should make them easy to use. You will find it worthwhile to master this chapter even if it needs more than one reading to do so.

6.1 Offsets and Coordinate Systems

If you made the FirstPart then you will already have met and used the ideas explained in this section.

When you Reference the PCNC, which should be done each time it is started, the X-, Y- and Z- axes are moved so that they are just off the home switches and the machine coordinates are set to 0.0. We saw that this is not useful when you want to run a part-program as the program will have its own view of where zero should be; in the case of FirstPart this is the top left corner of the material for X = 0, Y = 0 and the top surface for Z = 0. By moving the tool to this position and zeroing the axis DROs you defined relationship between the machine coordinates and the coordinates which the part-program would use – the Program Coordinates.

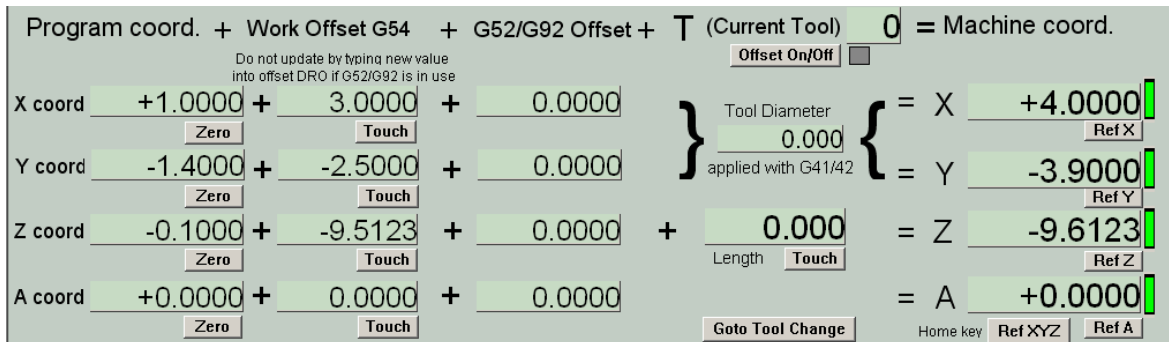


Figure 6.1 – Illustrating Program and Machine coordinates

This relationship is just a series of Offsets. The offsets are numbers which are added by the Control Software to your program X, Y, Z coordinates to get the machine coordinates.

Figure 6.1 is a view of part of the Offsets screen. You will probably find it useful to look at this screen on your machine as you read this material. The machine is set-up in Inch (imperial units). If your machine is set-up for metric units then you can either experiment with the same numbers as we use or, of course, roughly convert them to inches

To obtain figure 6.1 the PCNC was referenced (notice the green “referenced” LEDs at right of screen) and a tool jogged to the surface of a piece of stock just touching the surface near the

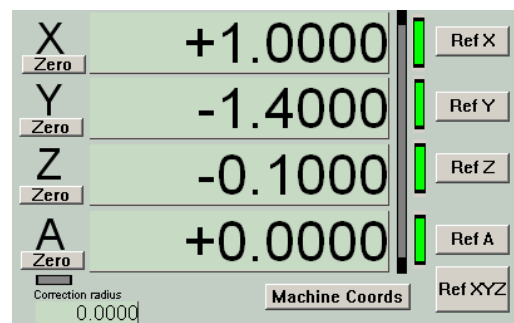


Figure 6.2 – Axis DROs corresponding to figure 6.1

top left corner. So you can imagine it, the jogging moved the Z-axis down about 9.5", the tool 3" to the right and 2.5" towards the front of the machine.

The *Zero* button for X-, Y- and Z-axes was clicked (we could have typed 0 into the axis DROs with exactly the same effect). The Manual Data Input (MDI) was then used to go to the program coordinate position X = 1, Y = -1.4 and Z = -0.1 by:

```
G00 X1.0 Y1.4 Z-0.1
```

Figure 6.2 shows the axis DROs that you would see, while the whole story is shown in figure 6.1. Let us use X as an example. The program coordinate for the tool is currently X= 1.0 (we did a G00 X1.0). The offset to be added to this to get the machine coordinate is 3". Adding these two values together gives the machine coordinate value of 4. These numbers are nice round values because we only roughly jogged to the top left corner of the stock.

Now look at Z. The jog was about 9.5" down but we put the tool exactly on the surface of the stock. The program coordinate is Z = -0.1 (from the G00 Z-0.1). The Z offset value is actually -9.5123 (so the jog of "about" 9.5" was actually 9.5123). Adding the program coordinate to the offset gives the machine coordinate of -9.6123. This is of course the distance that the head currently is below the home switch position.

The offsets we have been setting are called Work Offsets as they are different for each piece of work loaded onto the machine. Later we will see how Tool Offsets can allow for different projection of different tools from the spindle nose.

So to summarize, any move made in a part-program or MDI line which gives a new set of program coordinate values will cause the Control Software to add on the appropriate offsets and move the machine position to the newly calculated value of the machine coordinates value.

You may find it helpful to move the tool around on your machine above some scrap stock and use a tape-measure to reconcile the DRO readings with the machine position relative to the switches and the stock. When you have a clear picture in your mind of what is happening you have mastered the idea of offsets and should have no problems with this chapter.

6.2 Measuring Techniques

Now it is time to go from the theory of coordinate systems to the very practical choice of the best ways of finding where a tool is in relation to stock which you want to machine.

We will concentrate for the present on the position of the Z-axis. There are several reasons for this:

- The Z dimension is often the one with the highest accuracy constraints because the stock is often oversize in both dimensions in X and Y and is, at any event, not easy to locate precisely on the table.
- The Z direction is not only important for setting the initial position of the stock it needs to be accounted for when using several tools which are, almost inevitably, of different lengths.



Figure 6.3 – "Roll-your-own" gauge in use

6.2.1 "Roll-Your-Own" Gauge Method

This is a time honored machinists' method which traditionally used Rizla or similar cigarette paper as a gauge. The plastic foil

from candy wrapping is a good substitute. Both materials are fairly strong and around 0.002" in thickness.

Jog the tool **very** carefully down to the stock with a piece of paper/foil on it (figure 6.3). You will be able to feel when the paper/foil gets trapped.

You then know that the tool is 2 above the stock.

The advantage of this method is that you will always be able to find a gauge and it can be used on a very small flat surface on the stock (e.g., the boss of a casting). The disadvantage, particularly with hard stock and brittle tools (e.g., carbide) is that it is easy to jog down too far and damage the tool cutting edge.

6.2.2 Roller Gauge Method

For this method you need a short length of rod of a known diameter ($\frac{1}{4}$ " is quite suitable for most jobs). The shank of a twist-drill can be used in an emergency but take care to use one without scoring on it and measure the diameter rather than assuming it is the nominal size of the drill.

Jog the tool so it is clear of the stock but no more than the gauge diameter above it. You will be unable to roll the gauge under the tool (figure 6.4). While attempting to roll the gauge into the gap, very slowly jog upwards. You will easily feel the point where the gauge rolls under (figure 6.5). If you jog too far then it is easy to remove the gauge, jog down a little and try again.

You know that the tool is the diameter of the gauge above the stock.

The advantages of this method are that you do not risk crashing the tool into anything and, if you are repeatedly gauging, (say multiple tools) because the setting is always the same, you do not have to keep typing its size into the Control Program. The disadvantage is that you need to be able to jog slowly to get an accurate result. You may find it convenient to set the Slow Jog rate to suit the gauging and use the *Shift* key as an override to get into the starting position.



Figure 6.4 – Tool much too low for gauge



Figure 6.5 – Gauge just rolls under tool



Figure 6.6 – Parallel adjusted to gap

6.2.3 Adjustable Parallel Method

You need an adjustable sliding parallel gauge.

Jog by eye so that the tool is about the nominal size of the parallel. Insert it in the gap between tool and stock, slide it to fill the gap and lock it off (figure 6.6).

Carefully remove the parallel and measure it with a micrometer or caliper (figure 6.7).

The advantage of this method is that you do not need any careful jogging. The disadvantages are the need to measure the parallel and then to accurately type the result, which will generally be an awkward number, into the DRO.

You can of course use a combination of these methods depending on what you have to hand on a job and which is most convenient.



Figure 6.7 – Measuring the parallel gauge

6.2.4 Comments on Accuracy

With practice you will be able to use these methods to measure to better than 0.001". It is, however, interesting to look at other issues which affect the accuracy of your work.

The PCNC is a very rigid machine and is fitted with precision anti-backlash ballscrews and laser aligned in the factory so is inherently very accurate. The home switches will operate at the same point with a repeatability of much better than 0.001". You do however need to consider the following points:

- You are measuring at one point on the stock. If its surface is not flat or it is not clamped exactly parallel to the table then the height at other places will be different.
- A change of 40°F in the temperature of a ballscrew (say between early morning in the winter and when the shop and machine has heated up in the afternoon) will amount to a difference of around 0.004" when the tool is near the table at the bottom of the Z-axis.

General Accuracy Issues

The minimum discrete position move is 0.0001", this is the resolution of motion. Machine accuracy is closely related to ballscrew accuracy. Our ballscrews are accurate to 0.0006" per foot, but considering all the other factors that come into play, we prefer to keep accuracy expectations to 0.0013" per foot. Repeatability will be better than 0.001" per foot.

Machining is a mix of science, skill and art. The caveat in stating accuracy and repeatability is that these factors depend on the techniques used by the machinist. A skilled machinist can often deliver accuracy that exceeds the accuracy specified by the machine builder, while an inexperienced machinist may have difficulty delivering the expected accuracy. With this understanding, we cannot tell you what accuracy you will be able to achieve in your own work.

While a machine tool may seem absolutely rigid, the truth of the matter is that everything has some elasticity. Related to elasticity is the compressibility of components such as ball nuts and bearings. Preloading of bearings and ballscrews can remove the physical open space between moving parts, but the technique cannot eliminate compressibility. The key to achieving maximum accuracy is understanding and controlling the magnitude and direction of forces. Maximum accuracy is achieved when the forces are minimized, as occurs in a finishing cut. Maximum repeatability is achieved when the forces are repeatable, both in magnitude and direction.

6.3 Setting Work Offsets

Now that we have a range of ways of measuring the position of a tool we can look at entering the information onto the Offsets screen.

6.3.1 Direct Entry to Axis DRO

All the measurement methods give the current position of the tool relative to the stock, assuming the face you have measured to is to be zero in the program coordinates, all you need to do is to type the value into the relevant axis DRO. You can do this on any screen.

For example after using the adjustable parallels and without moving the axis, you would enter 0.547 into the Z-axis DRO (figure 6.8).

The program coordinates say Z = 0.547 and the tool is 0.547" above the stock so, for example, G00 Z0.0 will move the tool so it exactly touches the stock.

You may have noticed that in making FirstPart we cheated a little by assuming the paper gauge was zero thickness. This is often sufficiently accurate and allows use of the Zero buttons rather than typing a value.

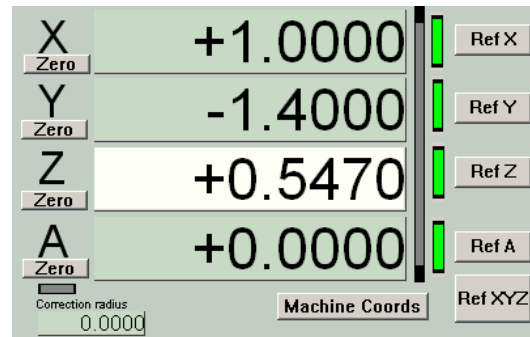


Figure 6.8 – Gauge size entered to DRO

6.3.2 Using the Touch Buttons

The “roll-you-own” and roller gauge methods will always give you the same measurement to be typed in. On the Offsets screen you can do this once and use it each time you want to set-up an offset.

The gauge thickness is typed into the Touch Correction DRO on the Offsets screen. Figure 6.9 shows this being set for the ¼" roller gauge. Note the correction is also enabled as shown by the LED by the On/Off button.

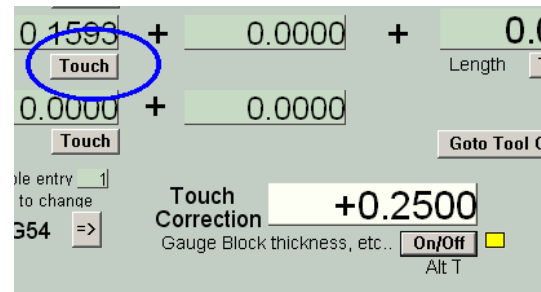


Figure 6.9 – Entering roller gauge size

When the gauge is in place, clicking the Touch button for the axis will set the offset. Figure 6.10 shows the program coordinates after Touch on Z-axis.

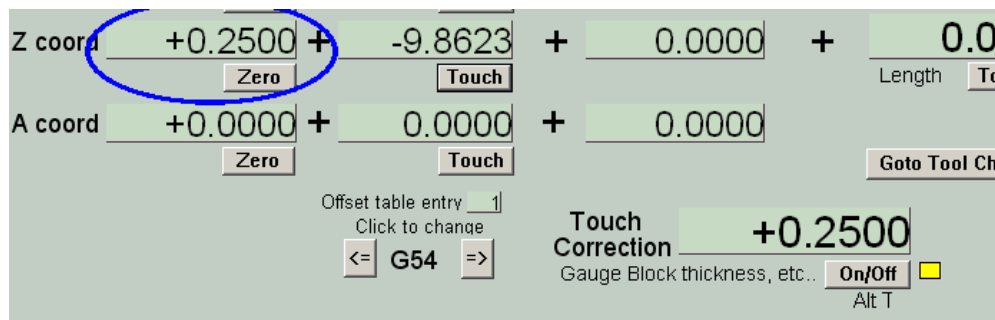


Figure 6.10 – Touch done on Z with roller gauge

6.3.3 Setting X and Y Offsets

It is sometimes quite acceptable to define the program coordinate zeros by eye as was done for FirstPart.

There are useful techniques when more accuracy is required.

6.3.3.1 With a Probe

By far the most accurate and often the quickest way of setting X and Y offsets to a piece of stock material or to a part-machined component is with a probe such as that illustrated in figure 6.11. Full details of the support for this accessory are given in chapter 8.

6.3.3.2 Measuring Off an Edge

Any of the above gauge methods can be used with a “tool” touching the edges of the stock or equally usefully the jaws of a vise used to hold the stock.

It is not really practicable to use a cutter because the spiral flutes do not give a clear place to gauge to. A cylindrical bar (e.g., ½" diameter) is ideal. It should be mounted in a tool holder so that it runs true. It is slightly easier to feel the contact point if a spherical finder is used. This is shown in the illustrations below.

Figure 6.12 show the measurement of the coordinates on the corner of the fixed jaw of a machine vise.

The value to be use in the *Touch Correction* DRO (or entered in the axis DRO) is half the diameter of the probe tool plus the measurement of the gauge. In the illustrated setup the ball has a diameter on 0.375" so with 0.002" foil the correction is numerically 0.1895".

The same technique can obviously be used on the edges of a piece of stock clamped to the table.

Note: You need to be careful about whether the correction is plus or minus.

In the case of the vise jaw it is negative (the center of the tool is actually nearer to 0.0 that the touching point). For the outside of a workpiece it should be positive. Thus in the example illustrated, the value typed into the Touch Correction DRO will be -0.1865 (figure 6.13).

6.3.3.3 Laser Centering Techniques

If you need to setup the X and Y program coordinates to features marked on, or already machined in, the workpiece then a laser finder or centering microscope is very useful.

Figure 6.14 shows a laser finder located near a center-punch mark on the stock. The point of light can be more clearly seen in the close-up view (figure 6.15)

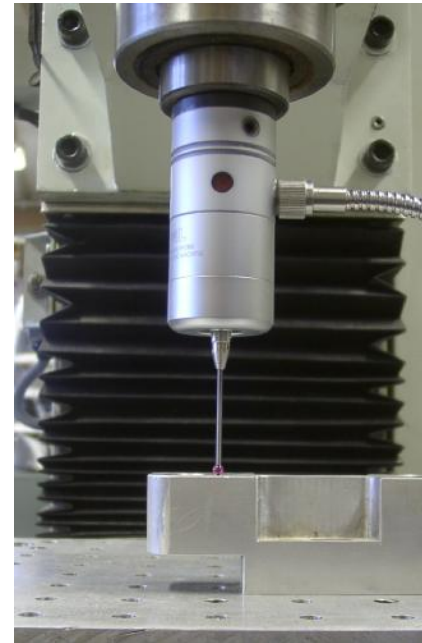


Figure 6.11 – The Tormach probe



Figure 6.12 – Measuring vise jaw Y position

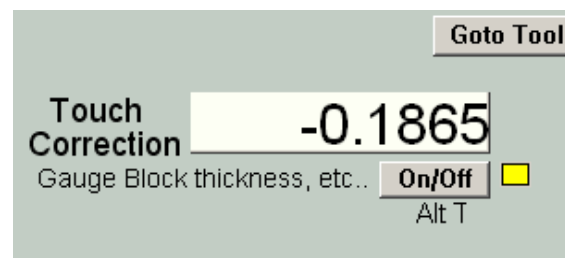


Figure 6.13 – “Inside” corrections are negative

Viewing the feature through a centering microscope has the same effect although the height of the Z-axis is then much more important as it affects the focusing of the microscope.

Obviously no correction is needed for the position as the beam is where the center of the tool will be.

6.4 Tool Offsets

If you only change tools very infrequently or if your tools are held in a collet chuck, which means that you cannot replace a given tool in exactly the same position each time it is inserted, then you will be able to do all your machining by using work offsets.

If, however, you need to change tools frequently within a job then you will find it very worthwhile to acquire sufficient tool holders so that there is one for each commonly used tool. Chapter 5 illustrates many of the available types of tool holder.

6.4.1 Tormach Tooling System

There are many advantages in basing your tool library on the Tormach Tooling System (TTS). It has a wide range of different styles of tool holder that are very reasonably priced and can be changed in the PCNC in a minimum time. Tools in TTS holders can be put into the PCNC spindle in exactly the same position each time. The Control Software can be set-up to record the projection (length) of each tool and allow for the differences whenever a tool is changed. This is done by another offset value, the Tool Offset, which is added into the conversion of a program Z coordinate to the corresponding machine Z coordinate.

If you use the TTS, then there is no difficulty in combining it with other tools in ordinary R8 holders. All the tool offset facilities remain available to you.

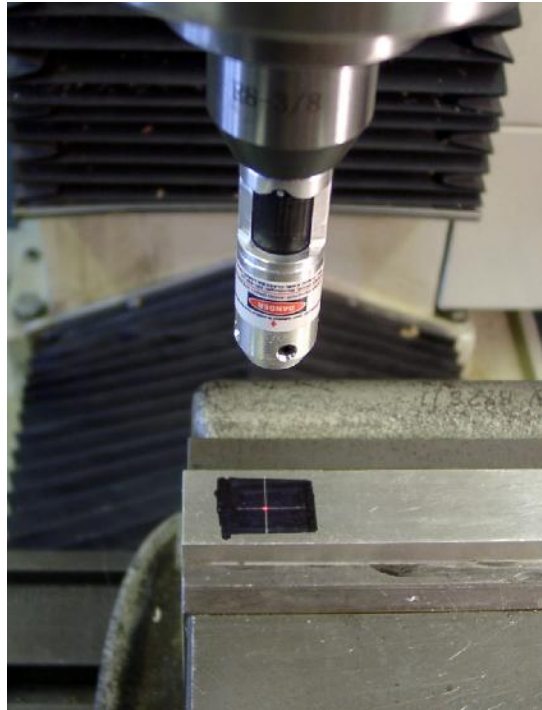


Figure 6.14 – Laser finder on scribed mark

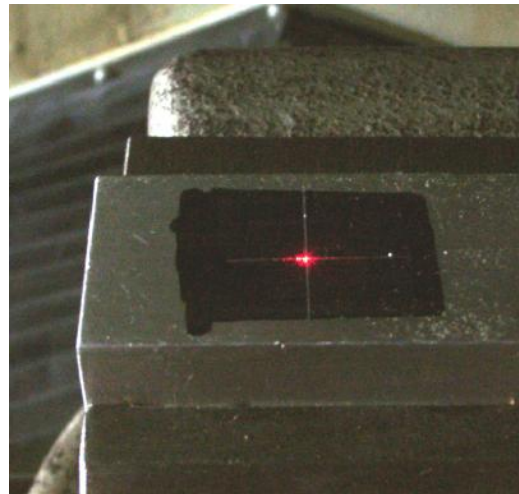


Figure 6.15 – Close-up of spot of light

6.4.2 How Tool Offsets Work

The principle of tool offsets is very simple. One tool is chosen to be the Master Tool. All other tool lengths are related to this, be they shorter or perhaps longer. For maximum accuracy of Z positions you will always use the master tool when setting the work offsets in the ways described above although, at the risk of some accumulated errors, you will realize that this can be done with any tool.

First we will look at how the offsets work in theory after some tools have been set-up then, in the following sections, see how to set them up with your tool collection.

Suppose there are three tools. Tool #1 has length 5.1", tool #2 is 3.2" and tool #3 is 3.3". These sizes should be easy to remember from the fractions. Do not worry for now where the lengths are measured from – if you like, assume it is from the front of the spindle nose.

We will assume that the machine has just been switched on and referenced and we will work on the Offsets screen. We jog the Z-axis down to a convenient position about 4" down from the home switch and put in tool #1. The Control Software has to know which tool is in use, so we

Program coord. + Work Offset G54 + G52/G92 Offset + T (Current Tool) 1 = Machine coord.

Do not update by typing new value into offset DRO if G52/G92 is in use

Offset On/Off ☐

X coord +0.0000 + 0.0000 + 0.0000 = X +0.0000

Y coord +0.0000 + 0.0000 + 0.0000 = Y +0.0000

Z coord -8.8794 + 0.0000 + 0.0000 + 5.100 = Z -3.7794

A coord +0.0000 + 0.0000 + 0.0000 = A +0.0000

Tool Diameter 0.500 applied with G41/42

Length 5.100

Goto Tool Change Home key Ref XYZ Ref A

Figure 6.16 – Offsets screen with tool 1 in place

enter 1 in the T (Current tool) DRO (figure 6.16).

What follows will be clearer if we set a work offset for this master tool so that at Z = 0.0 the tool touches the top of some stock. This is also what one would do to use the tool. So, using any of the tool touching methods, we set the work offset and use MDI to go to Z = 0.0 (figure 6.17).

Program coord. + Work Offset G54 + G52/G92 Offset + T (Current Tool) 1 = Machine coord.

Do not update by typing new value into offset DRO if G52/G92 is in use

Offset On/Off ☐

X coord +0.0000 + 0.0000 + 0.0000 = X +0.0000

Y coord +0.0160 + 0.0000 + 0.0000 = Y +0.0160

Z coord +0.0000 + -10.3547 + 0.0000 + 5.100 = Z -5.2547

A coord +0.0000 + 0.0000 + 0.0000 = A +0.0000

Tool Diameter 0.500 applied with G41/42

Length 5.100

Goto Tool Change Home key Ref XYZ Ref A

Offset table entry 1 Click to change

Touch Correction +0.2500 Gauge Block thickness, etc. On/Off Alt T

Jog ON/OFF Ctrl-Alt-J

Jog Step - Num-* 0.0010 Step

Jog Mode Num-/ Cont. Step Slow Jog Rate

g0z-0.1 g0z-0.1 g00z0 g00z0

g00z0.0

Figure 6.17 – After setting the work offset with master tool and G00 to Z = 0.0

If you are observant you will see that the Z machine coordinate is lower because we jogged down to a gauge and then moved to Z = 0.0 in the program coordinates. You do not, however, generally have to bother about machine coordinates. They are displayed in case you want a cross-check of how the offsets are adding up.

Program coord. + Work Offset G54 + G52/G92 Offset + T (Current Tool) 2 = Machine coord.

Do not update by typing new value into offset DRO if G52/G92 is in use

Offset On/Off ☐

X coord +0.0000 + 0.0000 + 0.0000 = X +0.0000

Y coord +0.0160 + 0.0000 + 0.0000 = Y +0.0160

Z coord +1.9000 + -10.3547 + 0.0000 + 3.200 = Z -5.2547

A coord +0.0000 + 0.0000 + 0.0000 = A +0.0000

Tool Diameter 0.375 applied with G41/42

Length 3.200

Goto Tool Change Home key Ref XYZ Ref A

Offset table entry 1 Click to change

Touch Correction +0.2500 Gauge Block thickness, etc. On/Off Alt T

Jog ON/OFF Ctrl-Alt-J

Jog Step - Num-* 0.0010 Step

Jog Mode Num-/ Cont. Step Slow Jog Rate

g0z-0.1 g0z-0.1 g00z0 g00z0

g00z0.0

Figure 6.18 – Z-axis not moved but tool #2 selected

Next we put tool #2 in the spindle and enter 2 in the T DRO¹. Now the tool will not be touching the stock – it is after all 1.9" shorter than tool #1. Figure 6.18 shows that the Z program coordinate is, as expected, 1.9".

Executing G00 Z0.0 will move the Z-axis 1.9" down and, of course, put tool #2 in contact with the surface of the stock.

Finally it should be clear that jogging the Z-axis up so that there is room to put tool #3 in the spindle, selecting it in the T DRO and executing G00 Z0.0 again will put tool #3 in contact with the top of the stock.

The *Tool Diameter* is only used for the probe diameter for tool #99 and by the Control Software when it is implementing cutter radius compensation with G42 and G43. Generally this function is implemented within CAD/CAM software so the diameter information in the table is merely a reminder of the nature of the tool you should have loaded.

Remember that we did not mind where we measured the lengths of each tool from. It could be the spindle nose but it could be any point provided it is the **same** for each tool. The techniques below use an imaginary measuring point that makes the Master Tool have zero length.

You may find that you need to follow one of the setting procedures below to fully understand the theory. Choose the one that best matches the tooling you have.

6.4.3 Master Tool and Numbering Tools

You should choose one tool from your library to be the Master Tool. Ideally it should be a tool that you are very unlikely to ever take out of its tool holder. Try to avoid a tool that you are likely to break or will suffer a lot of wear. A four-flute end-mill is easier to gauge from than a two-flute end-cutting mill. If you have plenty of tool holders then it is ideal to use a non-cutting tool like the finder shown in figure 6.11 as master tool. If you choose your master tool as the longest of any tool you are likely to use and measure the tools in such a way that its length seems to be 0.0 then you will be protected from a risk of crashing a tool into the work if you inadvertently turn tool length offsets off. This may seem odd and all other tools will have negative length offsets but remember it does not matter where the lengths are measured from – only their relative lengths is important.

Load up the commonly used tools into their holders and number them. The master tool, if used for cutting, is conventionally 1 and others are numbered sequentially. It is often convenient that their position in a rack denotes their number.

If your master tool is never used for cutting then call it Tool #0

Draw up a grid on paper with four columns headed: Tool number, Description, Diameter and Relative Length. Above the grid leave a space marked. Actual Master Tool length.



Figure 6.19 – Measuring a TTS end-mill

¹ As an aside, we might actually have to jog up to get it in and then, before we change the T DRO, do G00 Z0.0 to put the Z-axis back in place it was with tool #1 – but it is as well to ignore this unless you are trying to follow this sequence on a machine.

6.4.4 Using the Probe and Tool Setter

If you have a probe kit including a tool setter sensor then the easiest way of loading the tool table is to use it probing with each tool in turn. This obviates any hand transcription of numbers. The equipment also simplifies accurate setting up of work in a vise or on the table. Details are given in Chapter 8.

6.4.5 Tool Length Offsets with TTS

You use the procedures in this section if you have the Tormach Tooling System and want to use its ability to preset tools off the PCNC machine.

If you have a TTS but are happy to set the tools in your PCNC then you can use the procedure in the next section to setup the lengths but still benefit from the ability to exchange tools very quickly.

If you have both TTS tools and conventional tool holders then you must mount your master tool in a TTS holder or must set-up the lengths of all tools using the procedure of the next section.

Fit the tools in their TTS holders and number them.

Zero the dial on the TTS height gauge onto the face of the surface plate by rotating its bezel.

Measure the height from the surface plate of the Master Tool and write this on the paper as Actual Master Tool Length (AMTL). Work in inches even if you will run the mill with metric tools and G-code.

Now put each tool in turn into the dummy spindle and use the height gauge to measure its length. Figure 6.19 shows this being done with an end-mill.

Enter the number, description, diameter and the measured length of each tool minus AMTL onto your paper table. The Relative Length will always be a negative number if your master tool is the longest.

When this is completed go to the Offsets screen of the Control Software and click the Edit/Save table Now button above the T DRO. Notice there is a similar button above the Work Offset column – do not use this one.

Enter the data from your paper table into the similar one on the screen. The values for the three example tools used above are shown in the table (figure 6.20).

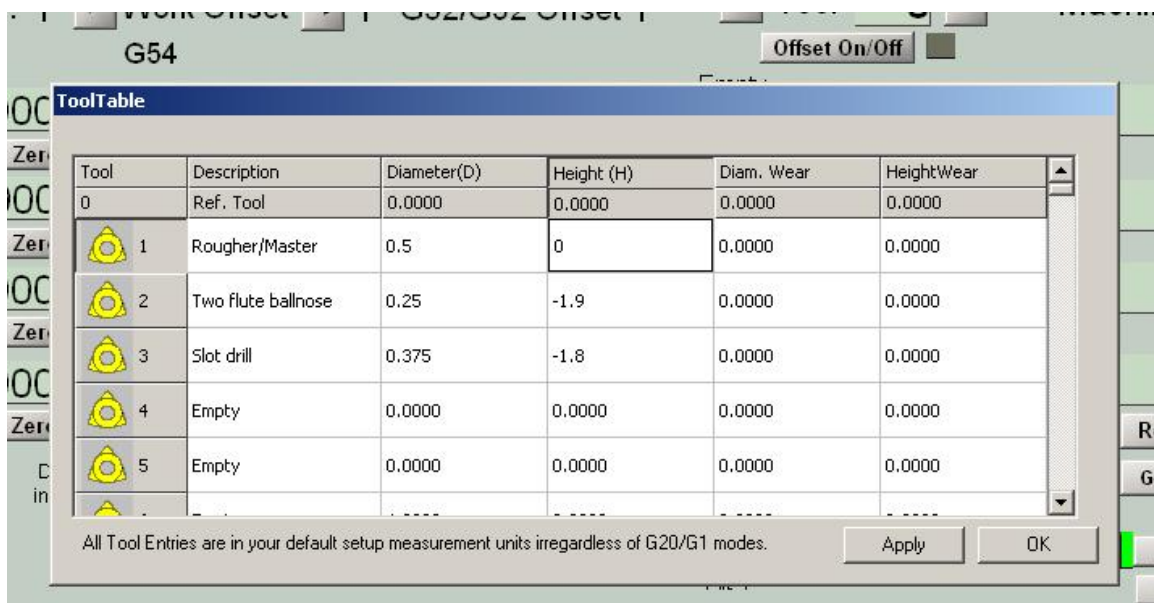


Figure 6.20 – Editing TTS data into the Tool Table

Finally click the Apply button followed by the OK button.

Your tool definitions will be stored permanently but can, of course, be edited at any time if you change the tooling in your holders.

Note: Some holders, like the drill chucks, can hold many tools and their lengths will be accurate enough even when they are interchanged. In this case you can enter more tools, with different numbers, than you have holders for; you will have to number the individual bits in their box.

6.4.6 Tool Length Offsets with General Tooling

With conventional tool holders you will not be able to measure them with a height gauge (unless, of course, you make your own dummy R8 spindle nose). The following procedure uses the PCNC as a measuring rig.

Mount a piece of flat stock on the table or in a vise on the PCNC. Work throughout on the Offsets screen.

If you have some TTS tools preset in the tool table then one of these **must** be the master tool.

Load the master tool, select tool #1 (or tool #0 if you have a non-cutting master) in the T DRO. If it is a TTS tool and so already in the table then its length should be shown as 0.0. If it is not in the table then type 0.0 in the Length press Enter to accept it.

Now set the work offset with it touching the work to Z = 0.0 by any of the gauging methods described above.

Now jog the Z-axis so you can load each tool holder in turn. Enter its number into the T DRO. Jog and gauge (by any of the touching methods already used) but instead of entering the gauged position into the Z-axis DRO you must always enter it into the Touch Correction DRO and click the tool Touch button marked Length – **not** any of those in the Work Offsets column. Enter the diameter of the tool in the Tool Diameter DRO. Do not forget that you have to press *Enter* to accept values typed into DROs. What you are doing is setting the lengths of all the tools relative to the Master Tool.

Figure 6.21 shows this having been done for tool #4.

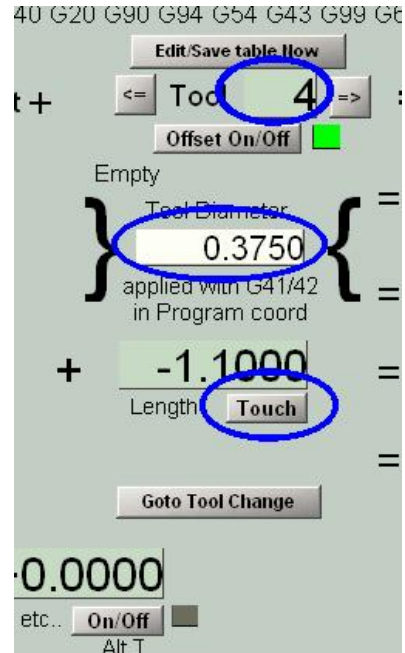


Figure 6.21 – Setting up a conventional tool holder as tool #4

6.4.7 Comments on Tool Offsets

If you do not save the Tool Table then, when you close down the Control Program, you will be asked if you want to save it. Unless you have made a serious error in setting up tool offsets you should save them or you will have to enter them again next time you run the Control Program.

Using the T DRO on a screen is a very easy way of changing a tool. The process is more complex in a part-program. This is because the RS-274 standard part-programs can run on large CNC systems with complex automatic tool changers. The RS-274 process consists of choosing the next tool to use with a T word, Loading the tool with M06 and applying its offsets with G43. The required commands will be generated by CAD/CAM systems and the Wizards. We strongly advise you to put all three stages (T~, M06 and G43 H~) on one line in your program and to check that the post processor of your CAM system does this. If you do not follow this "Triplet Rule" then you will encounter problems if you mix using tool lengths in the tool table and just setting the work offset when you change a tool.

6.5 Multiple Work Origins

6.5.1 G54 Work Offset

The previous explanations of offsets have only referred to a single set (X, Y, Z, A) of work offsets. These are always in effect when the Control Software is loaded.

It is sometimes useful to have another set of offsets and for them to be persistent when the system is closed down and reloaded. Suppose you have a machine vice permanently fixed to the table. It would be useful to remember the top left corner of its fixed jaw as this is a simple datum to locate stock.

6.5.2 Other Work Offsets

The control software allows you to setup a total of more than 250 different sets of work offsets; most people will only use the first six as these are easily selected using the codes G54 to G59. The Offsets screen has a pair of buttons to select the work offsets that are in use and displayed (figure 6.22).

When you *Touch* or enter values in the axis DROs then you will update the current set of work offsets.

You can save the work offset values using the *Edit/Save table Now* button above the Work Offset DROs. This button displays the complete table. It can be edited manually and is saved by clicking the *Save* button. The dialog is closed by clicking *OK*.

If you do not save the work offset table then, when you close down the Control Program, you will be asked if you want to save it. Unless you have made a serious error in setting up work offsets, you should save them or you will have to enter them again next time you run the Control Program. We advise you not to alter the G54 offsets although it is remembered from run to run of the Control Software.

Note: This is different from the Mach2 based version of the Control Software which always initialized G54 offsets to a fixed (usually zero) value.

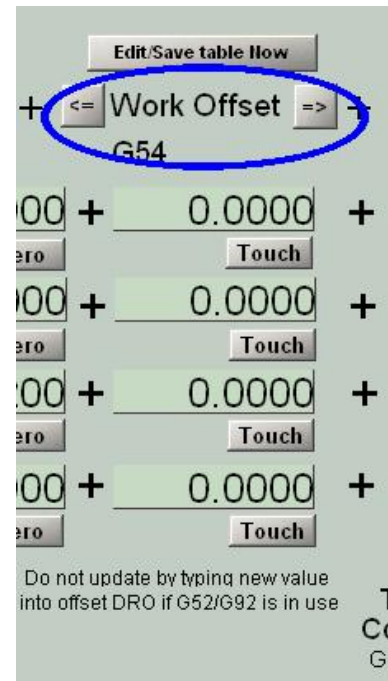


Figure 6.22 – Choosing Work Offsets

6.6 Cutter Diameter Compensation

6.6.1 CAD/CAM and Wizards

We have shown how it is necessary to allow for the diameter of the tool when cutting around an object or cutting out a pocket. You are strongly advised to do this by your CAD/CAM software or the Wizard which generates the part-program. The advantage of it being done this way is that the software has knowledge of the future path of the tool so can avoid overshooting on sharp inside corners. *You are unlikely to need to read the remainder of this chapter.*

6.6.2 Control Software Compensation

The Control Software does, however, implement compensation for the tool diameter using G41 and G42. This section describes how this works. In industrial applications this is aimed at allowing for a cutter which, through regrinding, is not exactly the diameter of the tool assumed when the part program was written. The compensation can be enabled by the machine operator rather than requiring the production of another part program.

Of the face of it, the problem should be easy to solve. All you need to do is to offset the controlled point by an appropriate X and Y to allow for the tool radius. Simple trigonometry gives the distances depending on the angle the direction of cut makes to the axes.

In practice it is not quite so easy. There are several issues but the main one is that the machine has to set a Z position before it starts cutting and at that time it does not know the direction in which the tool is going to be moving. This problem is solved by providing “pre-entry moves” which take place in waste material of the part. These ensure that the compensation calculations can be done before the actual part outline is being cut. Choice of a path which runs smoothly into the part’s outline also optimizes the surface finish. An exit move is sometimes used to maintain the finish at the end of a cut.

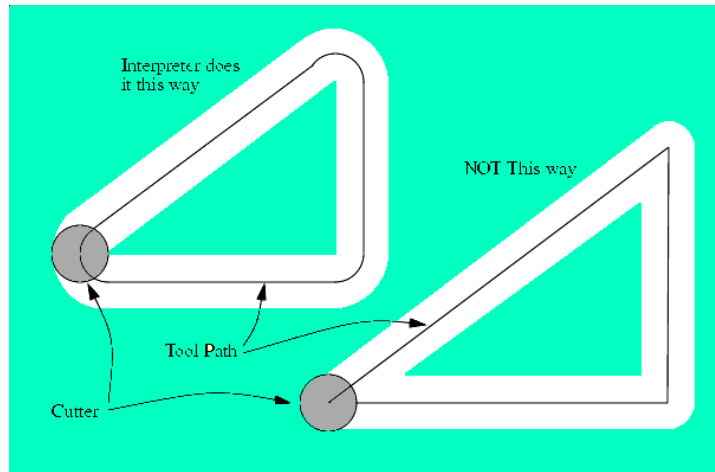


Figure 6.23 – Two possible toolpaths to cut triangle

6.6.3 Two Kinds of Contour

The Control Software handles compensation for two types of contour:

- The contour given in the part program code is the edge of material that is not to be machined away. We will call this type a “material edge contour.” This is the sort of code that might be “hand-written;”
- The contour given in the NC-code is the toolpath that would be followed by a tool of exactly the correct radius. We will call this type a “toolpath contour.” This is the sort of code that a CAD/CAM program might produce if it is aware of the intended cutter diameter.

The software does not have any setting that determines which type of contour is used, but the numerical description of the contour will, of course, differ (for the same part geometry) between the two types and the values for diameters in the tool table will be different for the two types.

6.6.3.1 Material Edge Contour

When the contour is the edge of the material, the outline of the edge is described in the part program. For a material edge contour, the value for the diameter in the tool table is the actual value of the diameter of the tool. The value in the table must be positive. The NC-code for a material edge contour is the same regardless of the (actual or intended) diameter of the tool.

Example 1:

Here is an NC program which cuts material away from the outside of the triangle (figure 6.23). In this example, the cutter compensation radius is the actual radius of the tool in use, which is 0.5. The value for the diameter in the tool table is twice the radius, which is 1.0.

```
N0010 G41 G1 X2 Y2 (turn compensation on and make entry move)
N0020 Y-1 (follow right side of triangle)
N0030 X-2 (follow bottom side of triangle)
N0040 X2 Y2 (follow hypotenuse of triangle)
N0050 G40 (turn compensation off)
```

This will result in the tool following a path consisting of an entry move and the path shown on the left going clockwise around the triangle. Notice that the coordinates of the triangle of

material appear in the NC-code. Notice also that the toolpath includes three arcs which are not explicitly programmed; they are generated automatically.

6.6.3.2 Toolpath Contour

When the contour is a toolpath contour, the path is described in the part program. It is expected that (except for during the entry moves) the path is intended to create some part geometry. The path may be generated manually or by a CAD/CAM program, considering the part geometry which is intended to be made. For the Control Software to work, the toolpath must be such that the tool stays in contact with the edge of the part geometry, as shown on the left side of figure 6.23. If a path of the sort shown on the right of figure 6.23 is used, in which the tool does not stay in contact with the part geometry all the time, the interpreter will not be able to compensate properly when undersized tools are used.

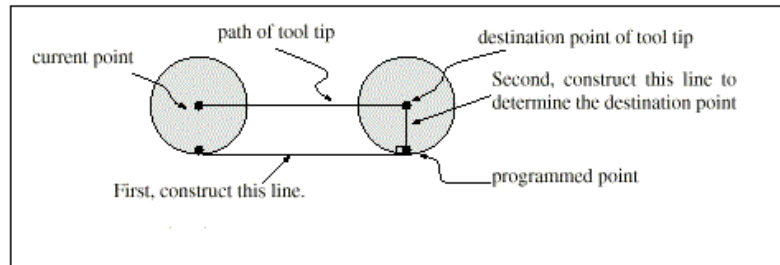


Figure 6.24 – First cutter compensation move – Straight

For a toolpath contour, the value for the cutter diameter in the tool table will be a small positive number if the selected tool is slightly oversized and will be a small negative number if the tool is slightly undersized. As implemented, if a cutter diameter value is negative, the interpreter compensates on the other side of the contour from the one programmed and uses the absolute value of the given diameter. If the actual tool is the correct size, the value in the table should be zero.

Toolpath Contour Example:

Suppose the diameter of the cutter currently in the spindle is 0.97 and the diameter assumed in generating the toolpath was 1.0; then the value in the tool table for the diameter for this tool should be -0.03. Here is an NC program which cuts material away from the outside of the triangle in the figure.

```
N0010 G1 X1 Y4.5 (make alignment move)
N0020 G41 G1 Y3.5 (turn compensation on and make first entry
                move)
N0030 G3 X2 Y2.5 I1 (make second entry move)
N0040 G2 X2.5 Y2 J-0.5 (cut along arc at top of toolpath)
N0050 G1 Y-1 (cut along right side of toolpath)
N0060 G2 X2 Y-1.5 I-0.5 (cut along arc at bottom right of tool
                path)
N0070 G1 X-2 (cut along bottom side of toolpath)
N0080 G2 X-2.3 Y-0.6 J0.5 (cut along arc at bottom left of
                toolpath)
N0090 G1 X1.7 Y2.4 (cut along hypotenuse of toolpath)
N0100 G2 X2 Y2.5 I0.3 J-0.4 (cut along arc at top of tool
                path)
N0110 G40 (turn compensation off)
```

This will result in the tool making an alignment move and two entry moves and then following a path slightly inside the path shown on the left in figure 6.24 going clockwise around the triangle. This path is to the right of the programmed path even though G41 was programmed, because the diameter value is negative.

First Move

The algorithm used for the first move when the first move is a straight line is to draw a straight line from the destination point which is tangent to a circle whose center is at the current point and whose radius is the radius of the tool. The destination point of the tool tip is then found as the center of a circle of the same radius tangent to the tangent line at the destination point (figure 6.24). If the programmed point is inside the initial cross section of the tool (the circle on the left), an error is signaled.

If the first move after cutter radius compensation has been turned on is an arc, the arc which is generated is derived from an auxiliary arc which has its center at the programmed center point, passes through the programmed end point and is tangent to the cutter at its current location. If the auxiliary arc cannot be constructed, an error is signaled. The generated arc moves the tool so that it stays tangent to the auxiliary arc throughout the move (figure 6.25).

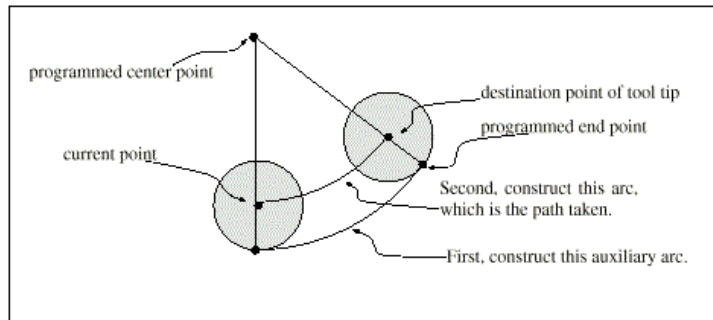


Figure 6.25 – First cutter compensation move – Arc

Regardless of whether the first move is a straight line or an arc, the Z-axis may also move at the same time. It will move linearly, as it does when cutter radius compensation is not being used. Rotary axis motions (A-, B- and C-axes) are allowed with cutter radius compensation, but using them would be very unusual. After the entry moves of cutter radius compensation, the interpreter keeps the tool tangent to the programmed path on the appropriate side. If a convex corner is on the path, an arc is inserted to go around the corner. The radius of the arc is half the diameter given in the tool table. When cutter radius compensation is turned off, no special exit move takes place.

The next move is what it would have been if cutter radius compensation had never been turned on and the previous move had placed the tool at its current position.

Programming Entry Moves

In general, an alignment move and two entry moves are needed to begin compensation correctly. However, where the programmed contour is a material edge contour and there is a convex corner on the contour, only one entry move (plus, possibly, a pre-entry move) is needed. The general method, which will work in all situations, is described first. We assume here that the programmer knows what the contour is already and has the job of adding entry moves.

General Method

The general method includes programming an alignment move and two entry moves. The entry moves given above will be used as an example. Here is the relevant code again:

```
N0010 G1 X1 Y4.5 (make alignment move to point C)
N0020 G41 G1 Y3.5 (turn compensation on and make first entry
                  move to point B)
N0030 G3 X2 Y2.5 I1 (make second entry move to point A)
```

Figure 6.26 shows the two entry moves but not the alignment move. First, pick a point A on the contour where it is convenient to attach an entry arc. Specify an arc outside the contour which begins at a point B and ends at A tangent to the contour (and going in the same direction as it is planned to go around the contour). The radius of the arc should be larger than half the diameter given in the tool table. Then extend a line tangent to the arc from B to some point C, located so that the line BC is more than one radius long. After the construction is finished, the code is written in the reverse order from the construction. Cutter radius compensation is turned on after

the alignment move and before the first entry move. In the code above, line N0010 is the alignment move, line N0020 turns compensation on and makes the first entry move and line N0030 makes the second entry move.

In this example, the arc AB and the line BC are fairly large, but they need not be. For a toolpath contour, the radius of arc AB need only be slightly larger than the maximum possible deviation of the radius of the tool from the exact size. Also for a toolpath contour, the side chosen for compensation should be the one to use if the tool is oversized. As mentioned earlier, if the tool is undersized, the interpreter will switch sides.

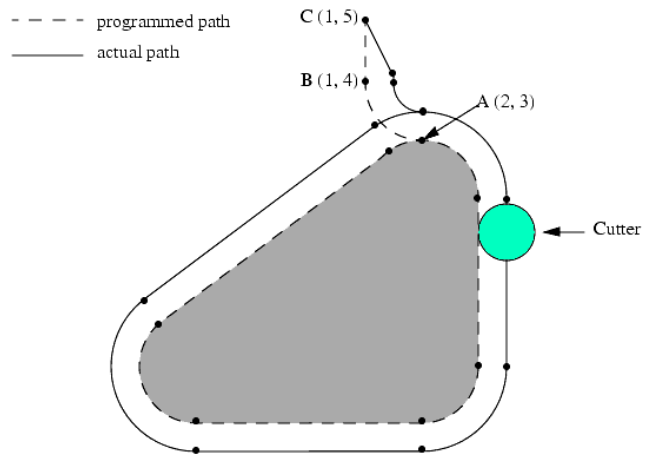


Figure 6.26 – The entry moves (omitting first alignment move)

Simple Method

If the contour is a material edge contour and there is a convex corner somewhere on the contour, a simpler method of making an entry is available (figure 6.27). First, pick a convex corner, D. Decide which way you want to go along the contour from D. In our example we are keeping the tool to the left of the contour and going next towards F. Extend the line FD (if the next part of the contour is an arc, extend the tangent to arc FD from D) to divide the area outside the contour near D into two regions. Make sure the center of the tool is currently in the region on the same side of the extended line as the material inside the contour near D. If not, move the tool into that region. In the example, point E represents the current location of the center of the tool. Since it is on the same side of line DF as the shaded triangle, no additional move is needed. Now write a line of NC-code that turns compensation on and moves to point D.

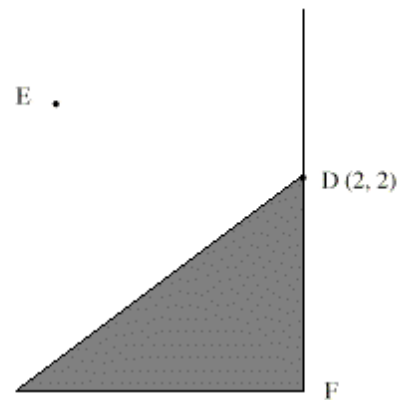


Figure 6.27 – Simple entry move

```
N0010 G41 G1 X2 Y2 (turn compensation on and make entry
move )
```

This method will also work at a concave corner on a toolpath contour, if the actual tool is oversized, but it will fail with a toolpath contour if the tool is undersized.

Note: We should re-iterate that it is much better for a CAM program to generate the path of the tool allowing for its diameter because it can "see" the implications of cutting complex corners in a way which the control software cannot.

7. Part-programming Language Reference

This section defines the language (G-codes, etc.) that are understood and interpreted by the Control Software.

This chapter is intended for reference purposes. If you want to learn about the principles of the control language so you can write programs by hand from first principles then you should consult an introductory textbook on G-code programming.

7.1 Definitions

7.1.1 *Control Software*

This is the term used for the program running under Microsoft Windows in the PC connected to your PNC-1100. In this chapter the Control Software is shortened to the CS.

7.1.2 *Linear Axes*

The X-, Y- and Z-axes form a standard right-handed coordinate system of orthogonal linear axes. Positions of the three linear motion mechanisms are expressed using coordinates on these axes.

7.1.3 *Rotational Axes*

The rotational axes are measured in degrees as wrapped linear axes in which the direction of positive rotation is counterclockwise when viewed from the positive end of the corresponding X-, Y- or Z-axis. By “wrapped linear axis,” we mean one on which the angular position increases without limit (goes towards plus infinity) as the axis turns counterclockwise and decreases without limit (goes towards minus infinity) as the axis turns clockwise. Wrapped linear axes are used regardless of whether or not there is a mechanical limit on rotation.

Clockwise or counterclockwise is from the point of view of the workpiece. If the workpiece is fastened to a turntable which turns on a rotational axis, a counterclockwise turn from the point of view of the workpiece is accomplished by turning the turntable in a direction that (for most common machine configurations) looks clockwise from the point of view of someone standing next to the machine.

7.1.4 *Scaling Input*

It is possible to set-up scaling factors for each axis. These will be applied to the values of X, Y, Z, A, I, J and R words whenever these are entered. This allows the size of features machined to be altered and mirror images to be created – by use of negative scale factors.

The scaling is the first thing done with the values and things like feed rate are always based on the scaled values.

The offsets stored in tool and fixture tables are not scaled before use. Scaling may, of course, have been applied at the time the values were entered (say using G10).

7.1.5 *Controlled Point*

The controlled point is the point whose position and rate of motion are controlled. When the tool length offset is zero (the default value), this is a point on the spindle axis (often called the gauge point) that is some fixed distance beyond the end of the spindle, usually near the end of a

tool holder that fits into the spindle. The location of the controlled point can be moved out along the spindle axis by specifying some positive amount for the tool length offset. This amount is normally the length of the cutting tool in use, so that the controlled point is at the end of the cutting tool.

7.1.6 *Coordinated Linear Motion*

To drive a tool along a specified path, a machining system must often coordinate the motion of several axes. We use the term “coordinated linear motion” to describe the situation in which, nominally, each axis moves at constant speed and all axes move from their starting positions to their end positions at the same time. If only the X-, Y- and Z-axes (or any one or two of them) move, this produces motion in a straight line, hence the word “linear” in the term. In actual motions, it is often not possible to maintain constant speed because acceleration or deceleration is required at the beginning and/or end of the motion. It is feasible, however, to control the axes so that, at all times, each axis has completed the same fraction of its required motion as the other axes. This moves the tool along the same path and we also call this kind of motion coordinated linear motion.

Coordinated linear motion can be performed either at the prevailing feed rate or at rapid traverse rate. If physical limits on axis speed make the desired rate unobtainable, all axes are slowed to maintain the desired path.

7.1.7 *Feed Rate*

The rate at which the controlled point or the axes move is nominally a steady rate which may be set by the user. In the Interpreter, the interpretation of the feed rate is as follows unless inverse time feed rate (G93) mode is being used:

- For motion involving one or more of the linear axes (X, Y, Z and optionally A, B, C), without simultaneous rotational axis motion, the feed rate means length units per minute along the programmed linear XYZ(ABC) path.
- For motion involving one or more of the linear axes (X, Y, Z and optionally A, B, C), with simultaneous rotational axis motion, the feed rate means length units per minute along the programmed linear XYZ(ABC) path combined with the angular velocity of the rotary axes multiplied by the appropriate axis Correction Diameter multiplied by pi, where $\pi = 3.14152$ (i.e., the declared “circumference” of the part).
- For motion of one rotational axis with X-, Y- and Z-axes not moving, the feed rate means degrees per minute rotation of the rotational axis.
- For motion of two or three rotational axes with X-, Y- and Z-axes not moving, the rate is applied as follows: Let dA, dB and dC be the angles in degrees through which the A-, B- and C-axes, respectively, must move. Let $D = \sqrt{dA^2 + dB^2 + dC^2}$. Conceptually, D is a measure of total angular motion, using the usual Euclidean metric. Let T be the amount of time required to move through D degrees at the current feed rate in degrees per minute. The rotational axes should be moved in coordinated linear motion so that the elapsed time from the start to the end of the motion is T plus any time required for acceleration or deceleration.

7.1.8 *Arc Motion*

Any pair of the linear axes (XY, YZ and XZ) can be controlled to move in a circular arc in the plane of that pair of axes. While this is occurring, the third linear axis and the rotational axes can be controlled to move simultaneously at effectively a constant rate. As in coordinated linear motion, the motions can be coordinated so that acceleration and deceleration do not affect the path.

If the rotational axes do not move, but the third linear axis does move, the trajectory of the controlled point is a helix.

The feed rate during arc motion is as described in Feed Rate above. In the case of helical motion, the rate is applied along the helix. Beware as other interpretations are used on other systems.

7.1.9 Coolant

Flood coolant and mist coolant may each be turned on independently. They are turned off together.

7.1.10 Dwell

A machining system may be commanded to dwell (i.e., keep all axes unmoving) for a specific amount of time. The most common use of dwell is to break and clear chips or for a spindle to get up to speed. The units in which you specify Dwell are either seconds or Milliseconds depending on the setting on Configure>Logic.

7.1.11 Units

Units used for distances along the X-, Y- and Z-axes may be measured in millimeters or inches. Units for all other quantities involved in machine control cannot be changed. Different quantities use different specific units. Spindle speed is measured in revolutions per minute. The positions of rotational axes are measured in degrees. Feed rates are expressed in current length units per minute, or in degrees per minute, as described above.

Warning: We advise you to check very carefully the system's response to changing units while tool and fixture offsets are loaded into the tables, while these offsets are active and/or while a part-program is executing

7.1.12 Current Position

The controlled point is always at some location called the "current position" and the Control Software always knows where that is. The numbers representing the current position are adjusted in the absence of any axis motion if any of several events take place:

- Length units are changed (see Warning above);
- Tool length offset is changed;
- Coordinate system offsets are changed.

7.1.13 Selected Plane

There is always a "selected plane," which must be the XY-plane, the YZ-plane or the XZ-plane of the machining system. The Z-axis is, of course, perpendicular to the XY-plane, the X-axis to the YZ-plane and the Y-axis to the XZ-plane.

7.1.14 Tool Table

Zero or one tool is assigned to each slot in the tool table.

7.1.15 Path Control Modes

The machining system may be put into any one of two path control modes: (1) exact stop mode, (2) constant velocity mode. In exact stop mode, the machine stops briefly at the end of each programmed move. In constant velocity mode, sharp corners of the path may be rounded slightly so that the feed rate may be kept up. These modes are to allow the user to control the compromise involved in turning corners because a real machine has a finite acceleration due to the inertia of its mechanism.

Exact stop does what it says. The machine will come to rest at each change of direction and the tool will therefore precisely follow the commanded path.

Constant velocity will overlap acceleration in the new direction with deceleration in the current one in order to keep the commanded feed rate. This implies a rounding of any corner but faster and smoother cutting. This is particularly important in routing and plasma cutting. The lower the acceleration of the machine axes, the greater will be the radius of the rounded corner.

In Plasma mode (set on *Configure Logic* dialog) the system attempts to optimize corner motion for plasma cutting by a proprietary algorithm.

It is also possible to define a limiting angle so that changes in direction of more than this angle will always be treated as Exact Stop even though Constant Velocity is selected. This allows gentle corners to be smoother but avoids excessive rounding of sharp corners even on machines with low acceleration on one or more axes. This feature is enabled in the *Configure Logic* dialog and the limiting angle is set by a DRO. This setting will probably need to be chosen experimentally depending on the characteristics of the machine tool and, perhaps, the toolpath of an individual job.

7.2 Interpreter Interaction with Controls

7.2.1 Feed and Speed Override controls

The CS has commands which enable (M48) or disable (M49) the feed and speed override switches. It is useful to be able to override these switches for some machining operations. The idea is that optimal settings have been included in the program and the operator should not change them.

7.2.2 Block Delete Control

If the block delete control is ON, lines of code which start with a slash (the block delete character) are not executed. If the switch is off, such lines are executed.

7.2.3 Optional Program Stop Control

The optional program stop control (see *Configure>Logic*) works as follows. If this control is ON and an input line contains an M1-code, program execution is stopped at the end of the commands on that line until the Cycle Start button is pushed.

7.3 Tool File

The CS maintains a tool file for each of the 256 tools which can be used.

Each data line of the file contains the data for one tool. This allows the definition of the tool length (Z-axis), tool diameter (for milling) and tool tip radius (for turning).

7.4 Part-programs Language

7.4.1 Overview

The language is based on lines of code. Each line (also called a “block”) may include commands to the machining system to do several different things. Lines of code may be collected in a file to make a program.

A typical line of code consists of an optional line number at the beginning followed by one or more “words.” A word consists of a letter followed by a number (or something that evaluates to a number). A word may either give a command or provide an argument to a command. For example, G1 X3 is a valid line of code with two words. “G1” is a command meaning “move in a straight line at the programmed feed rate,” and “X3” provides an argument value (the value of X should be 3 at the end of the move). Most commands start with either G (General) or M (Miscellaneous). The words for these commands are called “G-codes” and “M-codes.”

The language has two commands (M2 or M30), either of which ends a program. A program may end before the end of a file. Lines of a file that occur after the end of a program are not to be executed in the normal flow so will generally be parts of subroutines.

7.4.2 Parameters

The CS maintains an array of 10,320 numerical parameters. Many of them have specific uses. The parameters that are associated with fixtures are persistent over time. Other parameters will be undefined when The CS is loaded. The parameters are preserved when the interpreter is reset. Parameters 1 to 1000 can be used by the code of part-programs.

7.4.3 Coordinate Systems

The machining system has an absolute coordinate system and 254 work offset (fixture) systems.

You can set the offsets of tools by G10 L1 P~ X~ Z~ . The “P word” defines the tool number to be set.

You can set the offsets of the fixture systems using G10 L2 P~ X~ Y~ Z~ A~ B~ C~ . The P word defines the fixture to be set. The X, Y, Z, etc. words are the coordinates for the origin of the axes in terms of the absolute coordinate system.

You can select one of the first seven work offsets by using G54, G55, G56, G57, G58 or G59. Any of the 254 work offsets can be selected by G59 P~ (e.g., G59 P23 would select fixture 23). The absolute coordinate system can be selected by G59 P0.

You can offset the current coordinate system using G92 or G92.3. This offset will then be applied on top of work offset coordinate systems. This offset may be cancelled with G92.1 or G92.2.

You can make straight moves in the absolute machine coordinate system by using G53 with either G0 or G1.

7.5 Formatting Code Lines (Block)

A permissible line of input code consists of the following, in order, with the restriction that there is a maximum (currently 256) to the number of characters allowed on a line.

- Optional block delete character, which is a slash "/"
- Optional line number
- Any number of words, parameter settings and comments
- End of line marker (carriage return or line feed or both)

Any input not explicitly allowed is illegal and will cause the Interpreter to signal an error or to ignore the line.

Currently programs are limited to 999,999 lines of code.

Spaces and tabs are allowed anywhere on a line of code and do not change the meaning of the line, except inside comments. This makes some strange-looking input legal. For example, the line `G0X +0.1234Y 7` is equivalent to `G0 X+0.1234 Y7`

Blank lines are allowed in the input. They will be ignored.

Input is case insensitive, except in comments; thus, any letter outside a comment may be in upper or lower case without changing the meaning of a line.

7.5.1 Line Number

A line number is the letter N followed by an integer (with no sign) between 0 and 99,999,999 written without commas. Line numbers may be repeated or used out of order, although normal

practice is to avoid such usage. A line number is not required to be used (and this omission is common) but it must be in the proper place if it is used.

7.5.2 Subroutine Labels

A subroutine label is the letter O followed by an integer (with no sign) between 0 and 99999 written with no more than five digits (000009 is not permitted, for example). Subroutine labels may be used in any order but must be unique in a program although violation of this rule may not be flagged as an error. Nothing else except a comment should appear on the same line as a subroutine label. Note that line numbers are **not permitted** in the current release.

Letter	Meaning
A	A-axis of machine
B	B-axis of machine
C	C-axis of machine
D	tool radius compensation number
F	feed rate
G	general function (see Table 5)
H	tool length offset index
I	X-axis offset for arcs X offset in G87 canned cycle
J	Y-axis offset for arcs Y offset in G87 canned cycle
K	Z-axis offset for arcs Z offset in G87 canned cycle
L	number of repetitions in canned cycles/subroutines key used with G10
M	miscellaneous function (see Table 7)
N	line number
O	Subroutine label number
P	dwel time in canned cycles dwel time with G4 key used with G10 tapping depth in M871 – M874
Q	feed increment in G83 canned cycle repetitions of subroutine call
R	arc radius canned cycle retract level
S	spindle speed
T	tool selection
U	Synonymous with A
V	Synonymous with B
W	Synonymous with C
X	X-axis of machine
Y	Y-axis of machine
Z	Z-axis of machine

7.5.3 Word

A word is a letter other than N or O followed by a real value.

Words may begin with any of the letters (figure 7.1). The table includes N and O for completeness, even though, as defined above, line numbers are not words. Several letters (I, J, K, L, P and R) may have different meanings in different contexts.

A real value is some collection of characters that can be processed to come up with a number. A real value may be an explicit number (such as 341 or -0.8807), a parameter value, an expression or a unary operation value. Definitions of these follow immediately. Processing characters to come up with a number is called “evaluating.” An explicit number evaluates to itself.

7.5.3.1 Number

The following rules are used for (explicit) numbers. In these rules a digit is a single character between 0 and 9.

- A number consists of (1) an optional plus or minus sign, followed by (2) zero to many digits, followed, possibly, by (3) one decimal point, followed by (4) zero to many digits – provided that there is at least one digit somewhere in the number.
- There are two kinds of numbers: integers and decimals. An integer does not have a decimal point in it; a decimal does.
- Numbers may have any number of digits, subject to the limitation on line length. Only about seventeen significant figures will be retained, however (enough for all known applications).
- A non-zero number with no sign as the first character is assumed to be positive.

Notice that initial (before the decimal point and the first non-zero digit) and trailing (after the decimal point and the last non-zero digit) zeros are allowed but not required. A number written with initial or trailing zeros will have the same value when it is read as if the extra zeros were not there.

Numbers used for specific purposes by the Control Software are often restricted to some finite set of values or some to some range of values. In many uses, decimal numbers must be close to integers; this includes the values of indexes (for parameters and carousel slot numbers, for example), M-codes and G-codes multiplied by ten. A decimal number which is supposed to be close to an integer is considered close enough if it is within 0.0001 of an integer.

7.5.3.2 Parameter Value

A parameter value is the hash character # followed by a real value. The real value must evaluate to an integer between 1 and 10320. The integer is the parameter identification number. Think of it as the number of a pigeon-hole and distinguish this from its real *value* which is whatever number is stored in it.

The # character takes precedence over other operations, so that, for example, #1+2 means the number found by adding 2 to the value of parameter 1, not the value found in parameter 3. Of course, #[1+2] does mean the value found in parameter 3. The # character may be repeated; for example ##2 means the value of the parameter whose index is the (integer) value of parameter 2.

7.5.3.3 Expressions and Binary Operations

An expression is a set of characters starting with a left bracket ([) and ending with a balancing right bracket (]). In between the brackets are numbers, parameter values, mathematical operations and other expressions. An expression may be evaluated to produce a number. The expressions on a line are evaluated when the line is read, before anything on the line is executed. An example of an expression is:

```
[1+acos[0]-[#3**[4.0/2]]]
```

Binary operations appear only inside expressions. Nine binary operations are defined. There are four basic mathematical operations: addition (+), subtraction (-), multiplication (*) and division (/). There are three logical operations: non-exclusive or (OR), exclusive or (XOR) and logical and (AND). The eighth operation is the modulus operation (MOD). The ninth operation is the “power” operation (**) of raising the number on the left of the operation to the power on the right.

The binary operations are divided into three groups. The first group is: power. The second group is: multiplication, division and modulus. The third group is: addition, subtraction, logical non-exclusive or, logical exclusive or and logical. If operations are strung together (for example in the expression $[2.0 / 3 * 1.5 - 5.5 / 11.0]$), operations in the first group are to be performed before operations in the second group and operations in the second group before operations in the third group. If an expression contains more than one operation from the same group (such as the first / and * in the example), the operation on the left is performed first. Thus, the example is equivalent to: $[(2.0 / 3) * 1.5] - (5.5 / 11.0)$ which simplifies to $[1.0 - 0.5]$ which is 0.5.

The logical operations and modulus are to be performed on any real numbers, not just on integers. The number zero is equivalent to logical false and any non-zero number is equivalent to logical true.

7.5.3.4 Unary Operation Value

A unary operation value is either “ATAN” followed by one expression divided by another expression (for example $\text{ATAN}[2] / [1 + 3]$) or any other unary operation name followed by an expression (for example $\text{SIN}[90]$). The unary operations are: ABS (absolute value), ACOS (arc cosine), ASIN (arc sine), ATAN (arc tangent), COS (cosine), EXP (e raised to the given power), FIX (round down), FUP (round up), LN (natural logarithm), ROUND (round to the nearest whole number), SIN (sine), SQRT (square root) and TAN (tangent). Arguments to unary operations which take angle measures (COS, SIN and TAN) are in degrees. Values returned by unary operations which return angle measures (ACOS, ASIN and ATAN) are also in degrees.

The FIX operation rounds towards the left (less positive or more negative) on a number line, so that $\text{FIX}[2.8] = 2$ and $\text{FIX}[-2.8] = -3$, for example. The FUP operation rounds towards the right (more positive or less negative) on a number line; $\text{FUP}[2.8] = 3$ and $\text{FUP}[-2.8] = -2$, for example.

7.5.4 Parameter Setting

A parameter setting is the following four items one after the other:

- a pound character # ;
- a real value which evaluates to an integer between 1 and 10320;
- an equal sign = ;
- a real value.
- For example “#3 = 15” is a parameter setting meaning “set parameter 3 to 15.”

A parameter setting does not take effect until after all parameter values on the same line have been found. For example, if parameter 3 has been previously set to 15 and the line $\#3 = 6 \quad \text{G1} \times \#3$ is interpreted, a straight move to a point where x equals 15 will occur and the value of parameter 3 will be 6.

7.5.5 Comments and Messages

Printable characters and white space inside parentheses are considered a “comment.” A left parenthesis always starts a comment. The comment ends at the first right parenthesis found

thereafter. Once a left parenthesis is placed on a line, a matching right parenthesis must appear before the end of the line. Comments may not be nested; it is an error if a left parenthesis is found after the start of a comment and before the end of the comment. Here is an example of a line containing a comment: G80 M5 (stop motion)

An alternative form of comment is to use the two characters // . The remainder of the line is treated as a comment.

Comments do not cause the machining system to do anything.

A comment contains a message if MSG , appears after the left parenthesis and before any other printing characters. Variants of MSG , which include white space and lower case characters, are allowed. Note the comma which is required. The rest of the characters before the right parenthesis are considered to be a message to the operator. Messages are displayed on screen in the “Error” intelligent label.

7.5.6 *Item Repeats*

A line may have any number of G words, but two G words from the same modal group may not appear on the same line.

A line may have zero to four M words. Two M words from the same modal group may not appear on the same line.

For all other legal letters, a line may have only one word beginning with that letter.

If a parameter setting of the same parameter is repeated on a line, #3=15 #3=6, for example, only the last setting will take effect. It is silly, but not illegal, to set the same parameter twice on the same line.

If more than one comment appears on a line, only the last one will be used; each of the other comments will be read and its format will be checked, but it will be ignored thereafter. It is expected that putting more than one comment on a line will be very rare.

7.5.7 *Item Order*

The three types of item whose order may vary on a line (as given at the beginning of this section) are word, parameter setting and comment. Imagine that these three types of item are divided into three groups by type.

The first group (the words) may be reordered in any way without changing the meaning of the line.

If the second group (the parameter settings) is reordered, there will be no change in the meaning of the line unless the same parameter is set more than once. In this case, only the last setting of the parameter will take effect. For example, after the line #3=15 #3=6 has been interpreted, the value of parameter 3 will be 6. If the order is reversed to #3=6 #3=15 and the line is interpreted, the value of parameter 3 will be 15.

If the third group (the comments) contains more than one comment and is reordered, only the last comment will be used.

If each group is kept in order or reordered without changing the meaning of the line, then the three groups may be interleaved in any way without changing the meaning of the line. For example, the line g40 g01 #3=15 (so there!) #4=-7.0 has five items and means exactly the same thing in any of the 120 possible orders – such as #4=-7.0 g01 #3=15 g40 (so there!) – for the five items.

7.5.8 *Commands and Machine Modes*

The Control Software has many commands that cause a machining system to change from one mode to another. The mode stays active until some other command changes it implicitly or

explicitly. Such commands are called “modal.” For example, if coolant is turned on, it stays on until it is explicitly turned off. The G-codes for motion are also modal. If a G1 (straight move) command is given on one line, for example, it will be executed again on the next line if one or more axis words is available on the line, unless an explicit command is given on that next line using the axis words or canceling motion.

“Non-modal” codes have effect only on the lines on which they occur. For example, G4 (dwell) is non-modal.

7.6 Modal Groups

Modal commands are arranged in sets called “modal groups,” and only one member of a modal group may be in force at any given time. In general, a modal group contains commands for which it is logically impossible for two members to be in effect at the same time (e.g., measure in inches vs. measure in millimeters). A machining system may be in many modes at the same time, with one mode from each modal group being in effect (figure 7.2).

For several modal groups, when a machining system is ready to accept commands, one member of the group must be in effect. There are default settings for these modal groups. When the machining system is turned on or otherwise re-initialized, the default values are automatically in effect.

<p>The modal groups for G-codes are</p> <ul style="list-style-type: none"> • group 1 = {G00, G01, G02, G03, G38.2, G80, G81, G82, G84, G85, G86, G87, G88, G89} motion • group 2 = {G17, G18, G19} plane selection • group 3 = {G90, G91} distance mode • group 5 = {G93, G94} feed rate mode • group 6 = {G20, G21} units • group 7 = {G40, G41, G42} cutter radius compensation • group 8 = {G43, G49} tool length offset • group 10 = {G98, G99} return mode in canned cycles • group 12 = {G54, G55, G56, G57, G58, G59, G59.xxx} coordinate system selection • group 13 = {G61, G61.1, G64} path control mode
<p>The modal groups for M-codes are:</p> <ul style="list-style-type: none"> ◆ group 4 = {M0, M1, M2, M30} stopping ◆ group 6 = {M6} tool change ◆ group 7 = {M3, M4, M5} spindle turning ◆ group 8 = {M7, M8, M9} coolant (special case: M7 and M8 may be active at the same time) ◆ group 9 = {M48, M49} enable/disable feed and speed override controls
<p>In addition to the above modal groups, there is a group for non-modal G-codes:</p> <ul style="list-style-type: none"> ◆ group 0 = {G4, G10, G28, G30, G53, G92, G92.1, G92.2, G92.3}

Figure 7.2 – Modal groups

Group 1, the first group on the table, is a group of G-codes for motion. One of these is always in effect. That one is called the current motion mode.

It is an error to put a G-code from group 1 and a G-code from group 0 on the same line if both of them use axis words. If an axis word-using G-code from group 1 is implicitly in effect on a line (by having been activated on an earlier line) and a group 0 G-code that uses axis words appears on the line, the activity of the group 1 G-code is suspended for that line. The axis word-using G-codes from group 0 are G10, G28, G30 and G92.

The Control Software displays the current mode at the top of each screen.

7.7 G-codes

G-codes of the CS input language are shown in figure 7.3 and are described in more detail in this section.

The descriptions contain command prototypes, set in `courier` type.

In the command prototypes, the tilde (~) stands for a real value. As described earlier, a real value may be (1) an explicit number, 4.4, for example, (2) an expression, [2+2.4], for example, (3) a parameter value, #88, for example or (4) a unary function value, acos[0], for example.

In most cases, if axis words (any or all of X~, Y~, Z~, A~, B~, C~, U~, V~, W~) are given, they specify a destination point. Axis numbers relate to the currently active coordinate system, unless explicitly described as being in the absolute coordinate system. Where axis words are optional, any omitted axes will have their current value. Any items in the command prototypes not explicitly described as optional are required. It is an error if a required item is omitted.

U, V and W are synonyms for A, B and C. Use of A with U, B with V, etc. is erroneous (like using A twice on a line). In the detailed descriptions of codes U, V and W are not explicitly mentioned each time but are implied by A, B or C.

In the prototypes, the values following letters are often given as explicit numbers. Unless stated otherwise, the explicit numbers can be real values. For example, G10 L2 could equally well be written G[2*5] L[1+1]. If the value of parameter 100 were 2, G10 L#100 would also mean the same. Using real values which are not explicit numbers as just shown in the examples is rarely useful.

If L~ is written in a prototype the “~” will often be referred to as the “L number.” Similarly the “~” in H~ may be called the “H number,” and so on for any other letter.

If a scale factor is applied to any axis then it will be applied to the value of the corresponding X, Y, Z, A/U, B/V, C/W word and to the relevant I, J, K or R words when they are used.

7.7.1 Rapid Linear Motion – G00

(a) For rapid linear motion, program: G0 X~ Y~ Z~ A~ where all the axis words are optional, except that at least one must be used. The G00 is optional if the current motion mode is G0. This will produce coordinated linear motion to the destination point at the current traverse rate (or slower if the machine will not go that fast). It is expected that cutting will not take place when a G00 command is executing.

(b) If G16 has been executed to set a Polar Origin then for rapid linear motion to a point described by a radius and angle G0 X~ Y~ can be used. X~ is the radius of the line from the

Summary of G-codes	
G00	Rapid positioning
G01	Linear interpolation
G02	Clockwise circular/helical interpolation
G03	Counterclockwise circular/helical interpolation
G04	Dwell
G10	Coordinate system origin setting
G12	Clockwise circular pocket
G13	Counterclockwise circular pocket
G15/G16	Polar Coordinate moves in G00 and G01
G17	XY Plane select
G18	XZ plane select
G19	YZ plane select
G20/G21	Inch/millimeter unit
G28	Return home
G28.1	Reference axes
G30	Return home
G31	Straight probe
G40	Cancel cutter radius compensation
G41/G42	Start cutter radius compensation left/right
G43	Apply tool length offset (plus)
G49	Cancel tool length offset
G50	Reset all scale factors to 1.0
G51	Set axis data input scale factors
G52	Temporary coordinate system offsets
G53	Move in absolute machine coordinate system
G54	Use fixture offset 1
G55	Use fixture offset 2
G56-58	Use fixture offset 3, 4, 5
G59	Use fixture offset 6 / use general fixture number
G61/G64	Exact stop/Constant Velocity mode
G68/G69	Coordinate system rotation
G73	Canned cycle - peck drilling
G80	Cancel motion mode (including canned cycles)
G81	Canned cycle – drilling
G82	Canned cycle – drilling with dwell
G83	Canned cycle – peck drilling
G85	Canned cycle – boring, no dwell, feed out
G86	Canned cycle – boring, spindle stop, rapid out
G88	Canned cycle – boring, spindle stop, manual out
G89	Canned cycle – boring, dwell, feed out
G90	Absolute distance mode
G91	Incremental distance mode
G92	Offset coordinates and set parameters
G92.x	Cancel G92 etc.
G93	Inverse time feed mode
G94	Feed per minute mode
G95	Feed per rev mode
G98	Initial level return after canned cycles
G99	R-point level return after canned cycles

Figure 7-3. Table of G-codes

G16 polar origin and Y~ is the angle in degrees measured with increasing values counterclockwise from the 3 o'clock direction (i.e., the conventional four quadrant conventions).

Coordinates of the current point at the time of executing the G16 are the polar origin.

It is an error if all axis words are omitted.

If cutter radius compensation is active, the motion will differ from the above; see Cutter Compensation. If G53 is programmed on the same line, the motion will also differ; see Absolute Coordinates.

7.7.2 **Linear Motion at Feed Rate – G01**

(a) For linear motion at feed rate (for cutting or not), program: G01 X~ Y~ Z~ A~, where all the axis words are optional, except that at least one must be used. The G01 is optional if the current motion mode is G01. This will produce coordinated linear motion to the destination point at the current feed rate (or slower if the machine will not go that fast). (b) If G16 has been executed to set a polar origin then linear motion at feed rate to a point described by a radius and angle G00 X~ Y~ can be used. X~ is the radius of the line from the G16 polar origin and Y~ is the angle in degrees measured with increasing values counterclockwise from the 3 o'clock direction (i.e., the conventional four quadrant conventions).

Coordinates of the current point at the time of executing the G16 are the polar origin.

It is an error if all axis words are omitted.

If cutter radius compensation is active, the motion will differ from the above; see Cutter Compensation. If G53 is programmed on the same line, the motion will also differ; see Absolute Coordinates.

7.7.3 **Arc at Feed Rate – G02 and G03**

A circular or helical arc is specified using either G02 (clockwise arc) or G03 (counterclockwise arc). The axis of the circle or helix must be parallel to the X-, Y- or Z-axis of the machine coordinate system. The axis (or, equivalently, the plane perpendicular to the axis) is selected with G17 (Z-axis, XY-plane), G18 (Y-axis, XZ-plane) or G19 (X-axis, YZ-plane). If the arc is circular, it lies in a plane parallel to the selected plane.

If a line of code makes an arc and includes rotational axis motion, the rotational axes turn at a constant rate so that the rotational motion starts and finishes when the XYZ motion starts and finishes. Lines of this sort are hardly ever programmed.

If cutter radius compensation is active, the motion will differ from the above; see Cutter Compensation.

Two formats are allowed for specifying an arc. We will call these the center format and the radius format. In both formats the G02 or G03 is optional if it is the current motion mode.

7.7.3.1 **Radius Format Arc**

In the radius format, the coordinates of the end point of the arc in the selected plane are specified along with the radius of the arc. Program: G02 X~ Y~ Z~ A~ R~ (or use G03 instead of G02). R is the radius. The axis words are all optional except that at least one of the two words for the axes in the selected plane must be used. The R number is the radius. A positive radius indicates that the arc turns through 180 degrees or less, while a negative radius indicates a turn of 180 degrees to 359.999 degrees. If the arc is helical, the value of the end point of the arc on the coordinate axis parallel to the axis of the helix is also specified.

It is an error if:

- Both of the axis words for the axes of the selected plane are omitted;

- No R word is given;
- The end point of the arc is the same as the current point.

It is not good practice to program radius format arcs that are nearly full circles or are semicircles (or nearly semicircles) because a small change in the location of the end point will produce a much larger change in the location of the center of the circle (and, hence, the middle of the arc). The magnification effect is large enough that rounding error in a number can produce out-of-tolerance cuts. Nearly full circles are outrageously bad, semicircles (and nearly so) are only very bad. Other size arcs (in the range tiny to 165 degrees or 195 to 345 degrees) are OK.

Here is an example of a radius format command to mill an arc:

```
G17 G02 X 1.0 Y 1.5 R 2.0 Z 0.5
```

That means to make a clockwise (as viewed from the positive Z-axis) circular or helical arc whose axis is parallel to the Z-axis, ending where X=1.0, Y=1.5 and Z=0.5, with a radius of 2.0. If the starting value of Z is 0.5, this is an arc of a circle parallel to the XY-plane; otherwise it is a helical arc.

7.7.3.2 Center Format Arc

In the center format, the coordinates of the end point of the arc in the selected plane are specified along with the offsets of the center of the arc from the current location. In this format, it is OK if the end point of the arc is the same as the current point.

It is an error if when the arc is projected on the selected plane, the distance from the current point to the center differs from the distance from the end point to the center by more than 0.0002 inch (if inches are being used) or 0.002 millimeter (if millimeters are being used).

The center is specified using the I and J words. There are two ways of interpreting them. The usual way is that I and J are the center relative to the current point at the start of the arc. This is sometimes called *Incremental IJ mode*. The second way is that I and J specify the center as actual coordinates in the current system. This is rather misleadingly called *Absolute IJ mode*. The IJ mode is set using the button and LED on the Settings screen. The choice of modes is to provide compatibility with commercial controllers. You will probably find Incremental to be best. In Absolute it will, of course usually be necessary to use both I and J words unless by chance the arc's center is at the origin.

When the XY-plane is selected, program: G2 X~ Y~ Z~ A~ I~ J~ (or use G3 instead of G2). The axis words are all optional except that at least one of X and Y must be used. I and J are the offsets from the current location or coordinates – depending on IJ mode (X and Y directions, respectively) of the center of the circle. I and J are optional except that at least one of the two must be used. It is an error if:

- X and Y are both omitted;
- I and J are both omitted.

When the XZ-plane is selected, program: G02 X~ Y~ Z~ A~ I~ K~ (or use G03 instead of G02). The axis words are all optional except that at least one of X and Z must be used. I and K are the offsets from the current location or coordinates – depending on IJ mode (X and Z directions, respectively) of the center of the circle. I and K are optional except that at least one of the two must be used.

It is an error if:

- X and Z are both omitted;
- I and K are both omitted.

When the YZ-plane is selected, program: G02 X~ Y~ Z~ A~ J~ K~ (or use G03 instead of G02). The axis words are all optional except that at least one of Y and Z must be used. J and K are the offsets from the current location or coordinates – depending on IJ mode (Y and Z

directions, respectively) of the center of the circle. J and K are optional except that at least one of the two must be used.

It is an error if:

- Y and Z are both omitted;
- J and K are both omitted.

Here is an example of a center format command to mill an arc in Incremental IJ mode:

```
G17 G02 X1.0 Y1.6 I0.3 J0.4 Z0.9
```

That means to make a clockwise (as viewed from the positive Z-axis) circular or helical arc whose axis is parallel to the Z-axis, ending where X=1.0, Y=1.6 and Z=0.9, with its center offset in the X direction by 0.3 units from the current X location and offset in the Y direction by 0.4 units from the current Y location. If the current location has X=0.7, Y=0.7 at the outset, the center will be at X=1.0, Y=1.1. If the starting value of Z is 0.9, this is a circular arc; otherwise it is a helical arc. The radius of this arc would be 0.5.

The above arc in Absolute IJ mode would be:

```
G17 G02 X1.0 Y1.6 I1.0 J1.1 Z0.9
```

In the center format, the radius of the arc is not specified, but it may be found easily as the distance from the center of the circle to either the current point or the end point of the arc.

7.7.4 Dwell – G04

For a dwell, program: G04 P~. This will keep the axes unmoving for the period of time in seconds specified by the P number.

It is an error if the P number is negative.

7.7.5 Coordinate System Data Tool and Work Offset Tables – G10

See details of tool and work offsets for further information on coordinate systems.

To set the offset values of a tool, program:

G10 L1 P~ X~ Z~ A~, where the P number must evaluate to an integer in the range 0 to 255 – the tool number – and offsets of the tool specified by the P number are reset to the given. The A number will reset the tool tip radius. Only those values for which an axis word is included on the line will be reset. The Tool diameter cannot be set in this way.

To set the coordinate values for the origin of a fixture coordinate system, program:

G10 L2 P~ X~ Y~ Z~ A~, where the P number must evaluate to an integer in the range 1 to 255 – the fixture number (Values 1 to 6 corresponding to G54 to G59) – and all axis words are optional. The coordinates of the origin of the coordinate system specified by the P number are reset to the coordinate values given (in terms of the absolute coordinate system). Only those coordinates for which an axis word is included on the line will be reset.

It is an error if:

- The P number does not evaluate to an integer in the range 0 to 255.

If origin offsets (made by G92 or G92.3) were in effect before G10 is used, they will continue to be in effect afterwards.

The coordinate system whose origin is set by a G10 command may be active or inactive at the time the G10 is executed.

The values set will not be persistent unless the tool or fixture tables are saved using the buttons on Tables screen.

Example: G10 L2 P1 x3.5 y17.2 sets the origin of the first coordinate system (the one selected by G54) to a point where X is 3.5 and Y is 17.2 (in absolute coordinates). The Z

coordinate of the origin (and the coordinates for any rotational axes) are whatever those coordinates of the origin were before the line was executed.

7.7.6 ***Clockwise/Counterclockwise Circular Pocket – G12 and G13***

These circular pocket commands are a sort of canned cycle which can be used to produce a circular hole larger than the tool in use or with a suitable tool (like a woodruff key cutter) to cut internal grooves for “O” rings etc.

Program: G12 I~ for a clockwise move and G13 I~ for a counterclockwise move.

The tool is moved in the X direction by the value of the I word and a circle cut in the direction specified with the original X and Y coordinates as the center. The tool is returned to the center.

- Its effect is undefined if the current plane is not XY

7.7.7 ***Exit and Enter Polar Mode – G15 and G16***

It is possible for G0 and G1 moves in the X/Y plane only to specify coordinates as a radius and angle relative to a temporary center point; program G16 to enter this mode. The current coordinates of the controlled point are the temporary center.

Program: G15 to revert to normal Cartesian coordinates.

```
G0 X2.0 Y2.0          // normal G0 move to 2.0,2.0
G16 //start of polar mode.
G10 X1.0 Y45
( this will move to X = 2.7071, Y = 2.7071 which is a
spot on a circle) (of radius 1.0 at 45 degrees from
the initial coordinates of 2.0,2.0.)
```

This can be very useful, for example, for drilling a circle of holes. The code below moves to a circle of holes every 10 degrees on a circle of radius 5.0", center X = 1.0, Y = 5.5 and peck drills to Z = -0.6

```
G0 X1.0 Y5.5
G16
G1 X5.0 Y0           //polar move to a radius of 5.0 angle 0 deg
G83 Z-0.6            // peck drill
G1 Y10               // ten degrees from original center...
G83 Z-0.6
G1 Y20               // 20 degrees....etc...

G1 Y30

G1 Y40
> ...etc....
G15                  //back to normal Cartesian
```

Note:

- (1) You **must not** make X or Y moves other than by using G0 or G1 when G16 is active;
- (2) This G16 is different to a Fanuc implementation in that it uses the current point as the polar center. The Fanuc version requires a lot of origin shifting to get the desired result for any circle not centered on 0,0.

7.7.8 ***Plane Selection – G17, G18 and G19***

Program G17 to select the XY-plane, G18 to select the XZ-plane or G19 to select the YZ-plane. The effects of having a plane selected are discussed in under G2/3 and Canned cycles.

7.7.9 Length Units – G20 and G21

Program G20 to use inches for length units and program G21 to use millimeters.

It is usually a good idea to program either G20 or G21 near the beginning of a program before any motion occurs and not to use either one anywhere else in the program. It is the responsibility of the user to be sure all numbers are appropriate for use with the current length units.

7.7.10 Return to Home – G28 and G30

A home position is defined (by parameters 5161-5166). The parameter values are in terms of the absolute coordinate system, but are in unspecified length units.

To return to home position by way of the programmed position, program:

G28 X~ Y~ Z~ A~ (or use G30). All axis words are optional. The path is made by a traverse move from the current position to the programmed position, followed by a traverse move to the home position. If no axis words are programmed, the intermediate point is the current point, only one move is made.

7.7.11 Reference Axes – G28.1

Program: G28.1 X~ Y~ Z~ A~ to reference the given axes. The axes will move at the current feed rate towards the home switch(es), as defined by the Configuration. When the absolute machine coordinate reaches the value given by an axis word then the feed rate is set to that defined by Configure>Config Referencing. Provided the current absolute position is approximately correct, then this will give a soft stop onto the reference switch(es).

7.7.12 Straight Probe – G31

7.7.12.1 Straight Probe Command

Program: G31 X~ Y~ Z~ A~ to perform a straight probe operation. The rotational axis words are allowed, but it is better to omit them. If rotational axis words are used, the numbers must be the same as the current position numbers so that the rotational axes do not move. The linear axis words are optional, except that at least one of them must be used. The tool in the spindle must be a probe.

It is an error if:

- The current point is less than 0.01 inch (0.254 millimeter) from the programmed point;
- G31 is used in inverse time feed rate mode;
- Any rotational axis is commanded to move;
- No X-, Y- or Z-axis word is used.

In response to this command, the machine moves the controlled point (which should be at the end of the probe tip) in a straight line at the current feed rate toward the programmed point; if the probe trips, the probe decelerates.

After successful probing, parameters 2000 to 2005 will be set to the coordinates of the location of the controlled point at the time the probe tripped (not where it stopped) or if it does not trip to the coordinates at the end of the move and a triplet giving X, Y and Z at the trip will be written to the triplet file if it has been opened by the M40 macro/OpenDigFile() function (q.v.). Code in macros can determine if a point is a trip or just end of the move by inspecting if the DIGITIZE input is active after the G31 command.

7.7.12.2 Using the Straight Probe Command

Using the straight probe command, if the probe shank is kept nominally parallel to the Z-axis (i.e., any rotational axes are at zero) and the tool length offset for the probe is used, so that the controlled point is at the end of the tip of the probe:

- Without additional knowledge about the probe, the parallelism of a face of a part to the XY-plane may, for example, be found;
- If the probe tip radius is known approximately, the parallelism of a face of a part to the YZ or XZ-plane may, for example, be found;
- If the shank of the probe is known to be well-aligned with the Z-axis and the probe tip radius is known approximately, the center of a circular hole, may, for example, be found;
- If the shank of the probe is known to be well-aligned with the Z-axis and the probe tip radius is known precisely, more uses may be made of the straight probe command, such as finding the diameter of a circular hole.

If the straightness of the probe shank cannot be adjusted to high accuracy, it is desirable to know the effective radii of the probe tip in at least the +X, -X, +Y and -Y directions. These quantities can be stored in parameters either by being included in the parameter file or by being set in a part-program.

Using the probe with rotational axes not set to zero is also feasible. Doing so is more complex than when rotational axes are at zero and we do not deal with it here.

7.7.12.3 Example Code

As a usable example, see the code for finding the center and diameter of a circular hole. For this code to yield accurate results, the probe shank must be well-aligned with the Z-axis, the cross section of the probe tip at its widest point must be very circular and the probe tip radius (i.e., the radius of the circular cross section) must be known precisely. If the probe tip radius is known only approximately (but the other conditions hold), the location of the hole center will still be accurate, but the hole diameter will not.

```

N010 (probe to find center and diameter of circular hole)
N020 (This program will not run as given here. You have to)
N030 (insert numbers in place of <description of number>.)
N040 (Delete lines N020, N030 and N040 when you do that.)
N050 G0 Z <Z-value of retracted position> F <feed rate>
N060 #1001=<nominal X-value of hole center>
N070 #1002=<nominal Y-value of hole center>
N080 #1003=<some Z-value inside the hole>
N090 #1004=<probe tip radius>
N100 #1005=[<nominal hole diameter>/2.0 - #1004]
N110 G0 X#1001 Y#1002 (move above nominal hole center)
N120 G0 Z#1003 (move into hole - to be cautious, substitute G1
for G0 here)
N130 G31 X[#1001 + #1005] (probe +X side of hole)
N140 #1011=#2000 (save results)
N150 G0 X#1001 Y#1002 (back to center of hole)
N160 G31 X[#1001 - #1005] (probe -X side of hole)
N170 #1021=[[#1011 + #2000] / 2.0] (find pretty good X-value of
hole center)
N180 G0 X#1021 Y#1002 (back to center of hole)
N190 G31 Y[#1002 + #1005] (probe +Y side of hole)
N200 #1012=#2001 (save results)
N210 G0 X#1021 Y#1002 (back to center of hole)
N220 G31 Y[#1002 - #1005] (probe -Y side of hole)
N230 #1022=[[#1012 + #2001] / 2.0] (find very good Y-value of
hole center)
N240 #1014=[#1012 - #2001 + [2 * #1004]] (find hole diameter in
Y-direction)

```

```

N250 G0 X#1021 Y#1022 (back to center of hole)
N260 G31 X[#1021 + #1005] (probe +X side of hole)
N270 #1031=#2000 (save results)
N280 G0 X#1021 Y#1022 (back to center of hole)
N290 G31 X[#1021 - #1005] (probe -X side of hole)
N300 #1041=[[#1031 + #2000] / 2.0] (find very good X-value of
hole center)
N310 #1024=[[#1031 - #2000 + [2 * #1004]]] (find hole diameter in
X-direction)
N320 #1034=[[#1014 + #1024] / 2.0] (find average hole diameter)
N330 #1035=[#1024 - #1014] (find difference in hole diameters)
N340 G0 X#1041 Y#1022 (back to center of hole)
N350 M2 (that's all, folks)

```

In the above code, an entry of the form <description of a number> is meant to be replaced by an actual number that matches the description of number. After this section of code has executed, the X-value of the center will be in parameter 1041, the Y-value of the center in parameter 1022 and the diameter in parameter 1034. In addition, the diameter parallel to the X-axis will be in parameter 1024, the diameter parallel to the Y-axis in parameter 1014 and the difference (an indicator of circularity) in parameter 1035. The probe tip will be in the hole at the XY center of the hole.

The example does not include a tool change to put a probe in the spindle. Add the tool change code at the beginning, if needed.

7.7.13 Cutter Radius Compensation – G40, G41 and G42

To turn cutter radius compensation off, program: G40. It is OK to turn compensation off when it is already off.

Cutter radius compensation may be performed only if the XY-plane is active.

To turn cutter radius compensation on left (i.e., the cutter stays to the left of the programmed path when the tool radius is positive), program: G41 D~. To turn cutter radius compensation on right (i.e., the cutter stays to the right of the programmed path when the tool radius is positive), program: G42 D~. The D word is optional; if there is no D word, the radius of the tool currently in the spindle will be used. If used, the D number should normally be the slot number of the tool in the spindle, although this is not required. It is OK for the D number to be zero; a radius value of zero will be used.

G41 and G42 can be qualified by a P-word. This will override the value of the diameter of the tool (if any) given in the current tool table entry.

It is an error if:

- The D number is not an integer, is negative or is larger than the number of carousel slots;
- The XY-plane is not active;
- Cutter radius compensation is commanded to turn on when it is already on.

The behavior of the machining system when cutter radius compensation is ON is described in the chapter on Cutter Compensation. Notice the importance of programming valid entry and exit moves.

7.7.14 Tool Length Offsets – G43, G44 and G49

To use a tool length offset, program: G43 H~, where the H number is the desired index in the tool table. It is expected that all entries in this table will be positive. The H number should be, but does not have to be, the same as the slot number of the tool currently in the spindle. The H

number may be zero; an offset value of zero will be used. Omitting H has the same effect as a zero value.

G44 is provided for compatibility and is used if entries in the table give negative offsets.

It is an error if the H number is not an integer, is negative or is larger than the number of carousel slots.

To use no tool length offset, program: G49 .

It is OK to program using the same offset already in use. It is also OK to program using no tool length offset if none is currently being used.

It is strongly advised to put the G43 command on the same line (block) as the T~ and the M06 which actually implements the change. If this is done then the control software anticipates the new offset during the time the operator has control for changing the tool. The operator can change the work Z offset safely if this condition is met.

7.7.15 *Scale Factors – G50 and G51*

To define a scale factor which will be applied to an X, Y, Z, A, I & J word before it is used program: G51 X~ Y~ Z~ A~ where the X, Y, Z etc. words are the scale factors for the given axes. These values are, of course, never themselves scaled.

It is **not** permitted to use unequal scale factors to produce elliptical arcs with G2 or G3.

To reset the scale factors of all axes to 1.0 program: G50 .

7.7.16 *Temporary Coordinate System Offset – G52*

To offset the current point by a given positive or negative distance (without motion), program: G52 X~ Y~ Z~ A~ , where the axis words contain the offsets you want to provide. All axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed.

It is an error if all axis words are omitted.

G52 and G92 use common internal mechanisms in the CS and may not be used together.

When G52 is executed, the origin of the currently active coordinate system moves by the values given.

The effect of G52 is cancelled by programming: G52 X0 Y0 etc.

Here is an example. Suppose the current point is at X=4 in the currently specified coordinate system, then G52 X7 sets the X-axis offset to 7 and so causes the X-coordinate of the current point to be -3.

The axis offsets are always used when motion is specified in absolute distance mode using any of the fixture coordinate systems. Thus all fixture coordinate systems are affected by G52.

7.7.17 *Move in Absolute Coordinates – G53*

For linear motion to a point expressed in absolute coordinates, program: G1 G53 X~ Y~ Z~ A~ (or similarly with G0 instead of G1), where all the axis words are optional, except that at least one must be used. The G0 or G1 is optional if it is in the current motion mode. G53 is not modal and must be programmed on each line on which it is intended to be active. This will produce coordinated linear motion to the programmed point. If G1 is active, the speed of motion is the current feed rate (or slower if the machine will not go that fast). If G0 is active, the speed of motion is the current traverse rate (or slower if the machine will not go that fast).

It is an error if:

- G53 is used without G0 or G1 being active;

- G53 is used while cutter radius compensation is on.

See relevant chapter for an overview of coordinate systems.

7.7.18 **Select Work Offset Coordinate System – G54 to G59 & G59 P~**

To select work offset #1, program: G54 and similarly for the first six offsets. The system-number-G-code pairs are: (1-G54), (2-G55), (3-G56), (4-G57), (5-G58), (6-G59).

To access any of the 254 work offsets (1 - 254) program: G59 P~ where the P word gives the required offset number. Thus G59 P5 is identical in effect to G58.

It is an error if one of these G-codes is used while cutter radius compensation is on.

See relevant chapter for an overview of coordinate systems.

7.7.19 **Set Path Control Mode – G61 and G64**

Program: G61 to put the machining system into exact stop mode or G64 for constant velocity mode. It is OK to program for the mode that is already active. These modes are described in detail above.

7.7.20 **Coordinate system rotation – G68 and G69**

A rotation transformation can be applied to the controlled point coordinates commanded by a part program or by the MDI line. To do this program G68 X~ Y~ R~ The X and Y words specify the center about which the rotation is to be applied in the current coordinate system. R is the angle of rotation in degrees with positive values being counter-clockwise.

If X or Y are omitted then zero is assumed. A and B can be used as synonyms for X and Y respectively.

To cancel rotation program G69. If a G68 is used while rotation is in operation a G69 is implied before it. In other words successive G68s are not cumulative and the X and Y points are always in an un-rotated system.

When a rotation is in use the X and Y axis DROs will be **red** to remind the operator that these values are program coordinate values which will be rotated.

This function can be used to compensate for work not exactly aligned on the table, to rotate the operation of a part program if it is coded with Y travel greater than X and so the work will not fit on the table or as software " vise soft-jaws".

Note:

- G68 may only be used in the XY plane (G17 mode)
- The effects of changing work offsets when a rotation transformation is in effect will be non-intuitive so it is wiser not to program this. Indeed care should be taken proving any program including transformations.
- There is very little standardization of the functions of this code across different CNC controls so careful checks should be made on code written for other machines.
- Jogging always takes place in the direction of the machine axes. The toolpath display frame is oriented to the physical axes and will show the part at the angle at which it will be cut.

7.7.21 **Canned Cycle – High Speed Peck Drill – G73**

The G73 cycle is intended for deep drilling or milling with chip breaking. See also G83. The retracts in this cycle break the chip but do not totally retract the drill from the hole. It is suitable for tools with long flutes which will clear the broken chips from the hole. This cycle takes a Q number which represents a "delta" increment along the Z-axis. Program:

G73 X~ Y~ Z~ A~ R~ L~ Q~

- Preliminary motion, as described in G81 to 89 canned cycles.
- Move the Z-axis only at the current feed rate downward by delta or to the Z position, whichever is less deep.
- Rapid back out by the distance defined in the G73 Pullback DRO on the Settings screen.
- Rapid back down to the bottom of the current hole, but backed off a bit.
- Repeat steps 1, 2 and 3 until the Z position is reached at step 1.
- Retract the Z-axis at traverse rate to clear Z.

It is an error if the Q number is negative or zero.

7.7.22 **Cancel Modal Motion – G80**

Program: G80 to ensure no axis motion will occur, to terminate canned cycles etc. Note that it cancels the current G0, G1, G2 or G3 mode so this must be re-established for the next move that is required. This particularly affects people adapting a CAM postprocessors from another machine as this behavior varies between different CNC controls.

It is an error if:

- Axis words are programmed when G80 is active, unless a modal group 0 G-code is programmed which uses axis words.

7.7.23 **Canned Cycles – G81 to G89**

The canned cycles G81 through G89 have been implemented as described in this section. Two examples are given with the description of G81 below.

All canned cycles are performed with respect to the currently selected plane. Any of the three planes (XY, YZ, and ZX) may be selected. Throughout this section, most of the descriptions assume the XY-plane has been selected. The behavior is always analogous if the YZ or XZ-plane is selected.

Rotational axis words are allowed in canned cycles, but it is better to omit them. If rotational axis words are used, the numbers must be the same as the current position numbers so that the rotational axes do not move.

All canned cycles use X, Y, R and Z numbers in the NC-code. These numbers are used to determine X, Y, R and Z positions. The R (usually meaning retract) position is along the axis perpendicular to the currently selected plane (Z-axis for XY-plane, X-axis for YZ-plane, Y-axis for XZ-plane). Some canned cycles use additional arguments.

For canned cycles, we will call a number “sticky” if, when the same cycle is used on several lines of code in a row, the number must be used the first time, but is optional on the rest of the lines. Sticky numbers keep their value on the rest of the lines if they are not explicitly programmed to be different. The R number is always sticky.

In incremental distance mode: when the XY-plane is selected, X, Y and R numbers are treated as increments to the current position and Z as an increment from the Z-axis position before the move involving Z takes place; when the YZ or XZ-plane is selected, treatment of the axis words is analogous. In absolute distance mode, the X, Y, R and Z numbers are absolute positions in the current coordinate system.

The L number is optional and represents the number of repeats. L=0 is not allowed. If the repeat feature is used, it is normally used in incremental distance mode, so that the same sequence of motions is repeated in several equally spaced places along a straight line. In absolute distance mode, L > 1 means “do the same cycle in the same place several times,” Omitting the L word is equivalent to specifying L=1. The L number is not sticky.

When L>1 in incremental mode with the XY-plane selected, the X and Y positions are determined by adding the given X and Y numbers either to the current X and Y positions (on the first go-around) or to the X and Y positions at the end of the previous go-around (on the repetitions). The R and Z positions do not change during the repeats.

The height of the retract move at the end of each repeat (called “clear Z” in the descriptions below) is determined by the setting of the retract mode: either to the original Z position (if that is above the R position and the retract mode is G98) or otherwise to the R position.

It is an error if:

- X, Y and Z words are all missing during a canned cycle;
- A P number is required and a negative P number is used;
- An L number is used that does not evaluate to a positive integer;
- Rotational axis motion is used during a canned cycle;
- Inverse time feed rate is active during a canned cycle;
- Cutter radius compensation is active during a canned cycle.

When the XY plane is active, the Z number is sticky and it is an error if:

- The Z number is missing and the same canned cycle was not already active;
- The R number is less than the Z number.

When the XZ plane is active, the Y number is sticky and it is an error if:

- The Y number is missing and the same canned cycle was not already active;
- The R number is less than the Y number.

When the YZ plane is active, the X number is sticky and it is an error if:

- The X number is missing and the same canned cycle was not already active;
- The R number is less than the X number.

7.7.23.1 Preliminary and In-Between Motion

At the very beginning of the execution of any of the canned cycles, with the XY-plane selected, if the current Z position is below the R position, the Z-axis is traversed to the R position. This happens only once, regardless of the value of L.

In addition, at the beginning of the first cycle and each repeat, the following one or two moves are made:

- A straight traverse parallel to the XY-plane to the given XY-position;
- A straight traverse of the Z-axis only to the R position, if it is not already at the R position.

If the XZ or YZ plane is active, the preliminary and in-between motions are analogous.

7.7.23.2 G81 Cycle

The G81 cycle is intended for drilling. Program: G81 X~ Y~ Z~ A~ R~ L~

- Preliminary motion, as described above.
- Move the Z-axis only at the current feed rate to the Z position.
- Retract the Z-axis at traverse rate to clear Z.

Example 1: Suppose the current position is (1, 2, 3) and the XY-plane has been selected and the following line of NC-code is interpreted.

```
G90 G81 G98 X4 Y5 Z1.5 R2.8
```

This calls for absolute distance mode (G90), old “Z” retract mode (G98) and calls for the G81 drilling cycle to be performed once. The X number and X position are 4. The Y number and Y position are 5. The Z number and Z position are 1.5. The R number and clear Z are 2.8. The following moves take place.

- a traverse parallel to the XY-plane to (4,5,3);
- a traverse parallel to the Z-axis to (4,5,2.8);
- a feed parallel to the Z-axis to (4,5,1.5);
- a traverse parallel to the Z-axis to (4,5,3).

Example 2: Suppose the current position is (1, 2, 3) and the XY-plane has been selected and the following line of NC-code is interpreted.

```
G91 G81 G98 X4 Y5 Z-0.6 R1.8 L3
```

This calls for incremental distance mode (G91), old “Z” retract mode and calls for the G81 drilling cycle to be repeated three times. The X number is 4, the Y number is 5, the Z number is -0.6 and the R number is 1.8. The initial X position is 5 (=1+4), the initial Y position is 7 (=2+5), the clear Z position is 4.8 (=1.8+3) and the Z position is 4.2 (=4.8-0.6). Old Z is 3.0

The first move is a traverse along the Z-axis to (1,2,4.8), since old Z < clear Z.

The first repeat consists of 3 moves.

- a traverse parallel to the XY-plane to (5,7,4.8);
- a feed parallel to the Z-axis to (5,7, 4.2);
- a traverse parallel to the Z-axis to (5,7,4.8).

The second repeat consists of 3 moves. The X position is reset to 9 (=5+4) and the Y position to 12 (=7+5).

- a traverse parallel to the XY-plane to (9,12,4.8);
- a feed parallel to the Z-axis to (9,12, 4.2);
- a traverse parallel to the Z-axis to (9,12,4.8).

The third repeat consists of 3 moves. The X position is reset to 13 (=9+4) and the Y position to 17 (=12+5).

- a traverse parallel to the XY-plane to (13,17,4.8);
- a feed parallel to the Z-axis to (13,17, 4.2);
- a traverse parallel to the Z-axis to (13,17,4.8).

7.7.23.3 G82 Cycle

The G82 cycle is intended for drilling. Program:

```
G82 X~ Y~ Z~ A~ R~ L~ P~
```

- Preliminary motion, as described above.
- Move the Z-axis only at the current feed rate to the Z position.
- Dwell for the P number of seconds.
- Retract the Z-axis at traverse rate to clear Z.

7.7.23.4 G83 Cycle

The G83 cycle (often called peck drilling) is intended for deep drilling or milling with chip breaking. See also G73. The retracts in this cycle clear the hole of chips and cut off any long stringers (which are common when drilling in aluminum). This cycle takes a Q number which represents a “delta” increment along the Z-axis. Program:

G83 X~ Y~ Z~ A~ R~ L~ Q~

- Preliminary motion, as described above.
- Move the Z-axis only at the current feed rate downward by delta or to the Z position, whichever is less deep.
- Rapid back out to the clear Z.
- Rapid back down to the current hole bottom, backed off a bit.
- Repeat steps 1, 2 and 3 until the Z position is reached at step 1.
- Retract the Z-axis at traverse rate to clear Z.

It is an error if:

- The Q number is negative or zero.

7.7.23.5 G85 Cycle

The G85 cycle is intended for boring or reaming, but could be used for drilling or milling.

Program: G85 X~ Y~ Z~ A~ R~ L~

- Preliminary motion, as described above.
- Move the Z-axis only at the current feed rate to the Z position.
- Retract the Z-axis at the current feed rate to clear Z.

7.7.23.6 G86 Cycle

The G86 cycle is intended for boring. This cycle uses a P number for the number of seconds to dwell. Program: G86 X~ Y~ Z~ A~ R~ L~ P~

- Preliminary motion, as described above.
- Move the Z-axis only at the current feed rate to the Z position.
- Dwell for the P number of seconds.
- Stop the spindle turning.
- Retract the Z-axis at traverse rate to clear Z.
- Restart the spindle in the direction it was going.

The spindle must be turning before this cycle is used. It is an error if:

- The spindle is not turning before this cycle is executed.

7.7.23.7 G88 Cycle

The G88 cycle is intended for boring. This cycle uses a P word, where P specifies the number of seconds to dwell. Program: G88 X~ Y~ Z~ A~ R~ L~ P~

- Preliminary motion, as described above.
- Move the Z-axis only at the current feed rate to the Z position.
- Dwell for the P number of seconds.
- Stop the spindle turning.
- Stop the program so the operator can retract the spindle manually.
- Restart the spindle in the direction it was going.

7.7.23.8 G89 Cycle

The G89 cycle is intended for boring. This cycle uses a P number, where P specifies the number of seconds to dwell. Program: G89 X~ Y~ Z~ A~ R~ L~ P~

- Preliminary motion, as described above.
- Move the Z-axis only at the current feed rate to the Z position.
- Dwell for the P number of seconds.
- Retract the Z-axis at the current feed rate to clear Z.

7.7.24 Distance Mode – G90 and G91

Interpretation of the CS-code can be in one of two distance modes: absolute or incremental.

To go into absolute distance mode, program: G90. In absolute distance mode, axis numbers (X, Y, Z, A) usually represent positions in terms of the currently active coordinate system. Any exceptions to that rule are described explicitly in this section describing G-codes.

To go into incremental distance mode, program: G91. In incremental distance mode, axis numbers (X, Y, Z, A) usually represent increments from the current values of the numbers.

I and J numbers always represent increments, regardless of the distance mode setting. K numbers represent increments.

7.7.25 G92 Offsets – G92, G92.1, G92.2 and G92.3

See the chapter on coordinate systems for full details. You are strongly advised not to use this legacy feature on any axis where there is another offset applied.

To make the current point have the coordinates you want (without motion), program:

G92 X~ Y~ Z~ A~, where the axis words contain the axis numbers you want. All axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed.

It is an error if all axis words are omitted.

G52 and G92 use common internal mechanisms in the CS and may not be used together.

When G92 is executed, the origin of the currently active coordinate system moves. To do this, origin offsets are calculated so that the coordinates of the current point with respect to the moved origin are as specified on the line containing the G92. In addition, parameters 5211 to 5214 are set to the X-, Y-, Z-, A-axis offsets. The offset for an axis is the amount the origin must be moved so that the coordinate of the controlled point on the axis has the specified value.

Here is an example. Suppose the current point is at X=4 in the currently specified coordinate system and the current X-axis offset is zero, then G92 X7 sets the X-axis offset to -3, sets parameter 5211 to -3 and causes the X-coordinate of the current point to be 7.

The axis offsets are always used when motion is specified in absolute distance mode using any of the fixture coordinate systems. Thus, all fixture coordinate systems are affected by G92.

Being in incremental distance mode has no effect on the action of G92.

Non-zero offsets may already be in effect when the G92 is called. They are in effect discarded before the new value is applied. Mathematically the new value of each offset is $A+B$, where A is what the offset would be if the old offset were zero and B is the old offset. For example, after the previous example, the X-value of the current point is 7. If G92 X9 is then programmed, the new X-axis offset is -5, which is calculated by $[(7-9) + -3]$. Put another way the G92 X9 produces the same offset whatever G92 offset was already in place.

To reset axis offsets to zero, program: G92 . 1 or G92 . 2 G92.1 sets parameters 5211 to 5214 to zero, whereas G92.2 leaves their current values alone.

To set the axis offset values to the values given in parameters 5211 to 5214, program: G92 . 3

You can set axis offsets in one program and use the same offsets in another program by programming G92 in the first program. This will set parameters 5211 to 5214. Do not use

G92.1 in the remainder of the first program. The parameter values will be saved when the first program exits and restored when the second one starts up. Use G92.3 near the beginning of the second program. That will restore the offsets saved in the first program.

7.7.26 Feed Rate Mode – G93, G94 and G95

Three feed rate modes are recognized: inverse time, units per minute and units per revolution of spindle. Program: G93 to start the inverse time mode (this is very infrequently employed).

Program: G94 to start the units per minute mode. Program: G95 to start the units per rev mode.

In inverse time feed rate mode, an F word means the move should be completed in [one divided by the F number] minutes. For example, if the F number is 2.0, the move should be completed in half a minute.

In units per minute feed rate mode, an F word on the line is interpreted to mean the controlled point should move at a certain number of inches per minute, millimeters per minute or degrees per minute, depending upon what length units are being used and which axis or axes are moving.

In units per rev feed rate mode, an F word on the line is interpreted to mean the controlled point should move at a certain number of inches per spindle revolution, millimeters per spindle revolution or degrees per spindle revolution, depending upon what length units are being used and which axis or axes are moving.

When the inverse time feed rate mode is active, an F word must appear on every line which has a G1, G2 or G3 motion and an F word on a line that does not have G1, G2 or G3 is ignored. Being in inverse time feed rate mode does not affect G0 (rapid traverse) motions.

It is an error if inverse time feed rate mode is active and a line with G1, G2 or G3 (explicitly or implicitly) does not have an F word.

7.7.27 Canned Cycle Return Level – G98 and G99

When the spindle retracts during canned cycles, there is a choice of how far it retracts:

1. Retract perpendicular to the selected plane to the position indicated by the R word;
2. Retract perpendicular to the selected plane to the position that axis was in just before the canned cycle started (unless that position is lower than the position indicated by the R word, in which case use the R word position).

To use option 1, program: G99 . To use option 2, program: G98 . Remember that the R word has different meanings in absolute distance mode and incremental distance mode.

7.8 Built-in M-codes

M-codes interpreted directly by the CS are shown in figure 7.5.

7.8.1 Program Stopping and Ending – M0, M1, M2 and M30

To stop a running program temporarily (regardless of the setting of the optional stop switch), program: M0.

To stop a running program temporarily (but only if the optional stop switch is on), program: M1.

It is OK to program M0 and M1 in MDI mode, but the effect will probably not be noticeable, because normal behavior in MDI mode is to stop after each line of input, anyway.

If a program is stopped by an M0, M1, pressing the cycle start button will restart the program at the following line.

To end a program, program: M2 or M30. M2 leaves the next line to be executed as the M2 line. M30 “rewinds” the G-code file. These commands can have the following effects depending on the options chosen on the Configure>Logic dialog:

- Axis offsets are set to zero (like G92.2) and origin offsets are set to the default (like G54).
- Selected plane is set to XY (like G17).
- Distance mode is set to absolute (like G90).
- Feed rate mode is set to Units per minute mode (like G94).
- Feed and speed overrides are set to ON (like M48).
- Cutter compensation is turned off (like G40).
- The spindle is stopped (like M5).
- The current motion mode is set to G1 (like G1).
- Coolant is turned off (like M9).

No more lines of code in the file will be executed after the M2 or M30 command is executed. Pressing cycle start will resume the program (M2) or start the program back at the beginning of the file (M30).

M-code	Meaning
M0	Program stop
M1	Optional program stop
M2	Program end
M3/4	Rotate spindle clockwise/counterclockwise
M5	Stop spindle rotation
M6	Tool change (by two macros)
M7	Mist coolant on
M8	Flood coolant on
M9	All coolant off
M30	Program end and Rewind
M47	Repeat program from first line
M48	Enable speed and feed override
M49	Disable speed and feed override
M98	Call subroutine
M99	Return from subroutine/repeat
M871, M872, M873, M874 ¹	Tapping Cycles
M998 ¹	Move to tool change position

Figure 7.4 – Built in M-codes

Note 1: These codes are “built in,” but application specific to the PCNC 1100

7.8.2 Spindle Control – M3, M4 and M5

To start the spindle turning clockwise at the currently programmed speed, program: M3.

To start the spindle turning counterclockwise at the currently programmed speed, program: M4.

For a PWM or Step/Dir spindle the speed is programmed by the S word. For an on/off spindle control it will be set by the gearing/pulleys on the machine.

To stop the spindle from turning, program: M5.

It is OK to use M3 or M4 if the spindle speed is set to zero; if this is done (or if the speed override switch is enabled and set to zero), the spindle will not start turning. If, later, the spindle speed is set above zero (or the override switch is turned up), the spindle will start turning. It is permitted to use M3 or M4 when the spindle is already turning or to use M5 when the spindle is already stopped but see the discussion on safety interlocks in configuration for the implications of a sequence that would reverse an already running spindle.

7.8.3 **Tool change – M6**

Provided tool change requests are not to be ignored (as defined in Configure>Logic), The CS will call a macro (q.v.) M6Start when the command is encountered. It will then wait for Cycle Start to be pressed, execute the macro M6End and continue running the part-program. You can provide Visual Basic code in the macros to operate your own mechanical tool changer and to move the axes to a convenient location for tool changing if you wish.

You are strongly advised to put the T~, the M06 and the G43 H~ on one line (block) of code. See G43 for more details.

7.8.4 **Coolant Control – M7, M8 and M9**

To turn mist coolant on, program: M7.

To turn flood coolant on, program: M8.

To turn all coolant off, program: M9.

It is always OK to use any of these commands, regardless of what coolant is on or off.

7.8.5 **Re-run from First Line – M47**

On encountering an M47 the part-program will continue running from its first line.

It is an error if M47 is executed in a subroutine.

The run can be stopped by the Pause or Stop buttons.

See also the use of M99 outside a subroutine to achieve the same effect.

7.8.6 **Override Control – M48 and M49**

To enable the speed and feed override, program: M48. To disable both overrides, program: M49. It is OK to enable or disable the switches when they are already enabled or disabled.

7.8.7 **Call Subroutine – M98**

To call a subroutine program: M98 P~ L~ or M98 ~P ~Q. The program must contain an O line with the number given by the P word of the Call. This O line is a sort of “label” which indicates the start of the subroutine. The O line may not have a line number (N word) on it. It, and the following code, will normally be written with other subroutines and follow an M2, M30 or M99 so it is not reached directly by the flow of the program.

The L word (or optionally the Q word) gives the number of times that the subroutine is to be called before continuing with the line following the M98. If the L (Q) word is omitted its value defaults to 1.

By using parameters values or incremental moves a repeated subroutine can make several roughing cuts around a complex path or cut several identical objects from one piece of material.

Subroutine calls may be nested. That is to say a subroutine may contain a M98 call to another subroutine. As no conditional branching is permitted it is not meaningful for subroutines to call themselves recursively.

7.8.8 *Return from Subroutine – M99*

To return from a subroutine program: M99. Execution will continue after the M98 which called the subroutine.

If M99 is written in the main program (i.e., not in a subroutine), then the program will start execution from the first line again. See also M47 to achieve the same effect.

7.9 *Application Defined M-codes*

7.9.1 *Self-reversing Tapping Cycles*

To use the Tormach Auto-reverse tapping heads program M871 P~ (or M872, M873, M874)

The P word specifies the depth to be threaded relative to the current Z position which will typically be just clear of the workpiece surface. The P word can be negative or positive with the same meaning.

Before use of these codes the size of tapping head to be used and the pitch of the thread must be defined in the appropriated place on the Settings screen. If the part program is running in Inch (G20) mode then the pitch is taken as a number of threads per inch. If it is metric (G21) then the pitch will be in millimeters. If the spindle speed is too high for the chosen pitch then an error message will be displayed and the cycle will not be performed.

The cycle operates as follows:

The currently set spindle speed and thread pitch are used to calculate the feed rate required to move the tap at the correct speed. The corresponding feedrate for the high speed retraction of the tap is also calculated. If this exceeds the available rapid rate then an error is displayed.

The tap is then fed downwards for the commanded depth (P word).

At the end of the down-feed the spindle is rapidly retracted by the appropriate distance for the size of the head in use. This engages the reverse drive.

The spindle is then retracted, at the higher reverse rate previously calculated, for a distance sufficient to ensure the tap springs clear of the hole.

The Z axis is then positioned at the original height above the work ready to move to another hole or another tool and operation.

Note:

- The above explanation is slightly simplified from the actual code used to aid understanding.
- For best results, especially for deep holes and blind tapping, the spindle speed chosen should be checked with a tachometer to ensure it is as near the commanded (S word) speed as possible.

7.9.2 *Goto Toolchange Position – M998*

Execution of M998 will send the machine to the tool change position. The tool change position is defined on the Settings screen. The Z-axis will move first, then X and Y. An entry of 9999 will disable the axis. Execution of this function requires the machine to be referenced (Homed).

7.9.3 *User Defined M-codes*

If any M-code is used which is not in the above list of built-in codes then the Control Software will attempt to find a file named “Mxx.m1S” in the Macros folder. If it finds the file then it will execute the Visual Basic script program it finds within it.

The Operator>Macros menu item displays a dialog which allows you to see the currently installed macros, to Load, Edit and Save or Save As the text. The dialog also has a Help button which will display the Visual Basic functions which can be called to control the Control Software. For example, you can interrogate the position of axes, move axes, interrogate input signals and control output signals.

New macros can be written using an external editor program like Notepad and saved in the Macros folder or you can load an existing macro within the Control Software, totally rewrite it and save it with a different file name.

7.10 Other Input Codes

7.10.1 Feed Rate – F

To set the feed rate, program: F~ .

Depending on the setting of the Feed Mode toggle the rate may be in units-per-minute or units-per-rev of the spindle.

The units are those defined by the G20/G21 mode.

Depending on the setting in Configure>Logic a revolution of the spindle may be defined as a pulse appearing on the Index input or be derived from the speed requested by the S word or Set Spindle speed DRO.

The feed rate may sometimes be overridden as described in M48 and M49 above.

7.10.2 Spindle Speed – S

To set the speed in revolutions per minute (rpm) of the spindle, program: S~ . The spindle will turn at that speed when it has been programmed to start turning. It is OK to program an S word whether the spindle is turning or not. If the speed override switch is enabled and not set at 100%, the speed will be different from what is programmed. It is OK to program S0; the spindle will not turn if that is done.

It is an error if: The S number is negative.

7.10.3 Select Tool – T

To select a tool, program: T~ , where the T number is slot number for the tool. The tool is not changed automatically. It is OK, but not normally useful, if T words appear on two or more lines with no tool change. It is OK to program T0; no tool will be selected. This is useful if you want the spindle to be empty after a tool change. It is an error if:

- A negative T number is used or a T number larger than 255 is used.

7.11 Order of Execution

The order of execution of items on a line is critical to safe and effective machine operation. Items are executed in the order shown in figure 7.5 if they occur on the same line.

If you wish to impose a different order (e.g. to turn coolant off before the spindle is stopped) just code the commands on separate blocks.

7.12 Error Handling

This section describes error handling in the Control Software.

The Control Software tends to ignore things that it does not understand. If a command does not work as expected or does not do anything, check that you have typed it correctly. Common mistakes are G0, instead of G0 (i.e., letter O instead of zero) and too many decimal points in

Order	Item
1	Comment (including message)
2	Set feed rate mode (G93, G94, G95)
3	Set feed rate (F)
4	Set spindle speed (S)
5	Select tool
6	Tool change (M6) and Execute M-code macros
7	Spindle On/Off (M3, M4, M5)
8	Coolant On/Off (M7, M8, M9)
9	Enable/disable overrides (M48, M49)
10	Dwell (G4)
11	Set active plane (G17, G18, G19)
12	Set length units (G20, G21)
13	Cutter radius compensation On/Off (G40, G41, G42)
14	Tool table offset On/Off (G43, G49)
15	Fixture table select (G54 – G58 & G59 P~)
16	Set path control mode (G61, G61.1, G64)
17	Set distance mode (G90, G91)
18	Set canned cycle return level mode (G98, G99)
19	Home, change coordinate system data (G10) or set offsets (G92, G94)
20	Perform motion (G0 to G3, G12, G13, G80 to G89 as modified by G53)
21	Stop or repeat (M0, M1, M2, M30, M47, M99)

Table 7.5 – Order of execution of commands on a line

numbers. The Control Software does not check for axis over travel (unless software limits are in use) or excessively high feeds or speeds. Nor does it does not detect situations where a legal command does something unfortunate, such as machining a fixture.

8. Machine Upgrades and Configuration

8.1 Fourth Axis – Rotary Table

The fourth axis can be used as a rotary table (i.e., with its axis of rotation parallel to the Z-axis) or more commonly as a dividing/indexing head with its axis of rotation parallel to the X-axis.

8.1.1 *Installing the Electronics*

Full details for installing the electronics for the optional rotary table are supplied in the form of a Service Bulletin with the installation kit.

8.1.2 *Utilizing the Fourth Axis*

8.1.2.1 Referencing and Zeroing the Fourth Axis

The Tormach rotary table does not have a built-in home switch. When you are setting it up, if the initial position is important, then you should use the + and - keys on the numeric keypad to jog it into position. When in a suitable position click Zero A. This will set the A DRO to zero in the current work offset system. If you use multiple work offsets then you should Zero A in all of them.

It is possible to provide a home switch in conjunction with a fixture on the fourth axis and interface it to the PCNC using the Accessory socket on the control panel. In this case the Ref A button can be used to move to this switch in the same way as the linear axes. If you do this accidentally, with no switch connected, then the movement can be cancelled with the Esc key.

8.1.2.2 Diameter Compensation Feature

To understand this it is useful to recall what happens on a G01 (line) move where two axes like X and Y are coordinated. The control software will attempt to move along the line at the feed rate requested by the F-word (or DRO). For example a move at a feed rate of 10 inches per minute from X = 0, Y = 0 to X = 1.0, Y = 1.0 will move both the X- and Y-axes at about 7 inches per minute. By Pythagoras' Theorem the diagonal speed is $\sqrt{7^2 + 7^2} = 9.9$.

Now if a move of X needs to be coordinated with one of A there is a problem because A moves in degrees. We only know how fast the tool is moving through the material if we also know the diameter of the part in the rotary axis. The Control Software will take this value from the Rotational Diameter DRO on the settings page.

Note: This feature only applies to coordinated moves. A single G01 move of the A-axis will use the feed rate defined by the F word as a number of degrees per minute.

8.1.3 *Fourth Axis Applications*

General use of a fourth axis requires CAD/CAM software that will create a part-program using the A-axis but useful work can be done with three axis software.



Figure 8.1 – Engraving on a cylinder

8.1.3.1 Engraving on a Periphery of a Cylinder

If you can engrave in X-Y with a Wizard or CAD/CAM software then it is easy to adapt an X-Y part-program to use the fourth axis (figure 8.1).

The rotary table is mounted with its axis parallel to the X-axis. The work to be engraved is mounted on a mandrel supported by the tailstock. It is of course important that it runs true or the depth of the engraving will be uneven.

The engraving is prepared as if it to be engraved in the X++ direction (i.e., the height of the letters is movement of Y). The code may not use G02 or G03 (arc) commands.

There are then three stages in engraving on the cylinder: (a) the part program is converted to be motion in A and X instead of X and Y; (b) the A motion is scaled to convert it to degrees; and (c) the radius of the cylinder is defined to ensure the correct feed rate on the surface being engraved.

Editing the Part-program

Load the X-Y engraving part-program for your text. Click the Edit button in the file control family. Use the Replace function of the edit to change all instances of “X” to “A” and then all instances of “Y” to “X.” This orients the engraving (figure 8.1).

Scaling the “Width” of the Text

The original part-program will move along the engraving in linear units (e.g., inches) but now it is edited to use degrees, the A-axis needs scaling. If the radius of the cylinder is R (mm or inches depending on what the control program is set in) then the A-axis scale should be set to $57.3 \div R$ (i.e., $360 / (2 \pi \times R)$). For example, the radius of the disk in figure 8.1 is 1.21" so the scale for A was set to 47.36 by typing this value into the A Scale DRO on the MDI screen (figure 8.2).

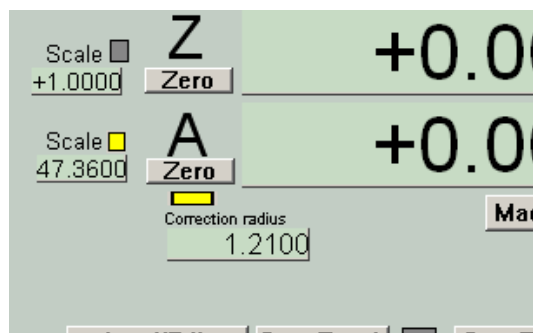


Figure 8.2 – Scale and Radius Correction

Setting the Radius Correction

So that the part-program's engraving feed rate is correct, the Radius correction DRO which is on Settings and also the MDI screen should also be set (figure 8.2).



Figure 8.3 – Gear cutting



Figure 8.4 – Gear cutting (tailstock removed for clarity)

8.1.3.2 Gear Cutting

Figures 8.3 and 8.4 show the fourth axis set-up for cutting a spur gear.

The G-code for this is easily produced by hand or by a Wizard as it consists of a series of passes cutting a tooth followed by indexing by an angle (say A) corresponding to the number of teeth (N). $A = 360 \div N$.

Machine upgrades and configuration

It is important that the gear cutter is exactly on the centerline of the gear and its mandrel. A steel rule can be used to set this quite accurately (figures 8.5 & 8.6).

The rule can be seen against the body of the chuck above the mandrel.

Notice that the test is more sensitive the smaller the diameter of the object onto which the rule is placed, so the setup is done on the mandrel not on the gear blank itself.



Figure 8.5— Ruler vertical so on center



Figure 8.6 – Ruler tilted to left at top so cutter is too high

If the gear cutter is mounted so that it can be inserted at the same height each time (e.g., on an R8 holder), then it may be worthwhile to set-up and save a work offset giving the centerline as $Y=0.0$ and $Z=0.0$ so that the setup does not need to be repeated each time the machine is switched on.

8.2 Digitizer Probes

8.2.1 Introduction to Uses of Probes and Tool Setters

If you mount a switch in the spindle instead of a cutting tool and connect the switch to the control computer so that the software can "read" it, then you can use the machine for measuring instead of cutting. The switch is usually arranged so it operates when a probe tip is moved horizontally in any direction or pushed up. Figure 8.7 shows a probe from the Tormach range of accessories (p/n 30668) probing the top of a partially machined part.

The Tormach probe has a standard TTS ($\frac{3}{4}$ ") shank and is best used in the $\frac{3}{4}$ " R8 collet (p/n 30146).

Note: We advise you to mark the spindle pulley with a dot of paint and always to insert the probe with the spindle in the same position. This eliminates any errors caused by random positioning. Keep the spindle lock ON when the probe is in. This will prevent

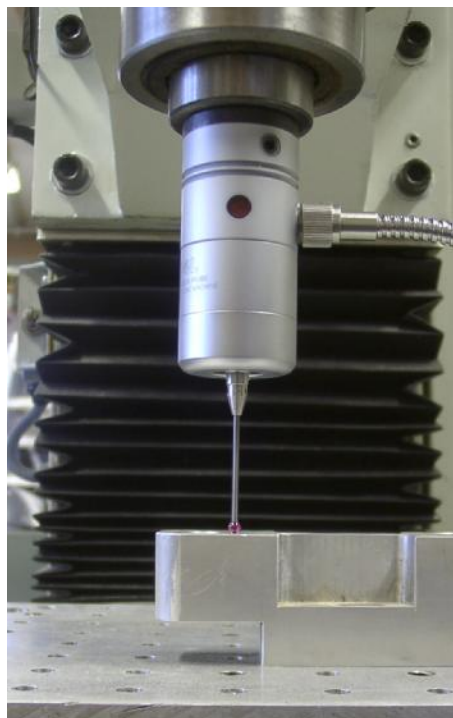


Figure 8.7 – Tormach probe

any accidental startup of the spindle motor which would rip the cable out of the probe body.

A Renishaw or other standard probe can be interfaced using an appropriate cable. Interface details are given below.

The other possibility is to have a tool in the spindle and "replace" the work with the switch. This allows measurement of the length of the tool and setting the coordinates of the plane on which the switch is standing. Figure 8.8 shows the Tormach Tool Setter (p/n 30669) in use on a tooling plate.

There are two main uses for the measurements taken:

(a) They can be used to set up the offsets in the Control Software so that the cutting defined by the part program is in the correct place relative to the stock material or part machined component

(b) They can be recorded to allow the shape of an existing model object to be input to a computer aided design (CAD) program.

We refer to the first application as **probing** and the second as **digitizing**.

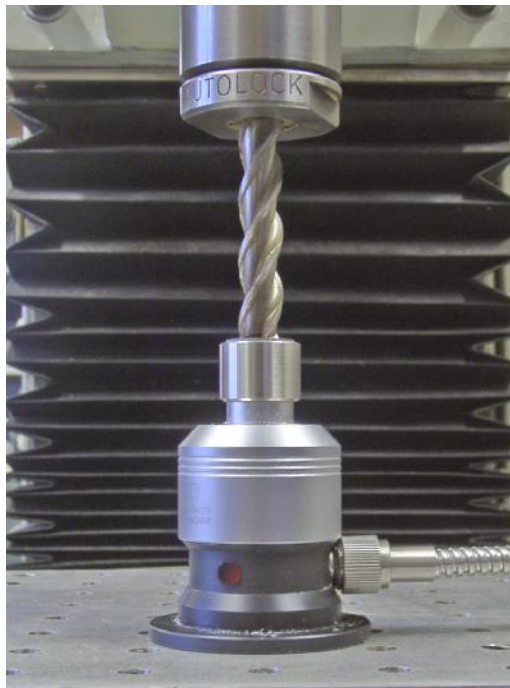


Figure 8.8 – Tormach toolsetter

8.2.2 Probing for Work/Tool Setting

The basic operation of the probe is controlled by the G31 code (q.v.). In practice you will not want to write programs to set up a job so a series of interactive screens is provided to perform probing operations. These are divided into measuring (probing) X and/or Y positions and probing Z positions. To make effective use of these screens you need a basic understanding of the concepts of Work Offsets and the Tool Length offset. You will have used Work Offsets already in setting up any work on the table even if only unconsciously by using the axis Zero buttons.

As they do not involve Tool length offsets we will describe the X/Y Probing screen first.

Before this there is one bit of configuration to do.

- Measure the diameter of your probe tip (usually made of ruby to avoid wear) with your best micrometer. The standard tip is 4 millimeter (0.1575")
- Enter this diameter for Tool #99 on the Offsets screen (figure 8.9). Ignore the Length for now and if it is zero then the Offset On/Off LED will not be lit. This is unimportant.

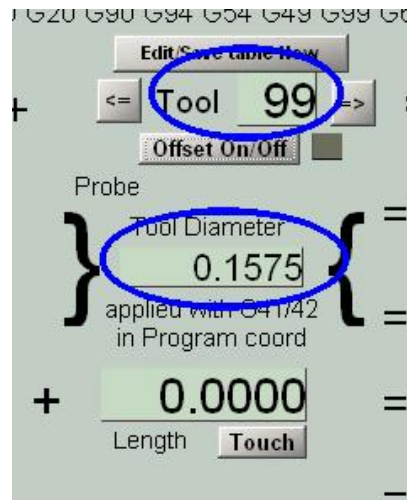


Figure 8.9 – Data for Probe (T#99)

8.2.2.1 Simple X/Y Probing

This screen (figure 8.10) allows you to set the origin (X = 0.0, Y = 0.0 point) to some feature of the stock material or a partly finished component.

The three pictures represent the three operations on this screen. (a) Finding center of a bore, (b) Finding a plane surface and (c) Finding the corner of the fixed jaw of a vise. The convention is on all screens is that a yellow screen button will make the PCNC move the probe looking for

the part. Obviously two things are necessary. The probe must be plugged in to the socket on the PCNC control panel (and hence be connected to the computer) and the Z height must be low enough so the probe tip will actually touch the work.

Important: If the probe is not properly connected then the Control Software will keep moving even when the probe touches the work. This can cause it to be bent. Each time that you connect up the probe you should check that the screen LED marked Test probe before use lights when you gently displace the tip. The red lights in the probe will illuminate too but the screen LED is a more complete test.

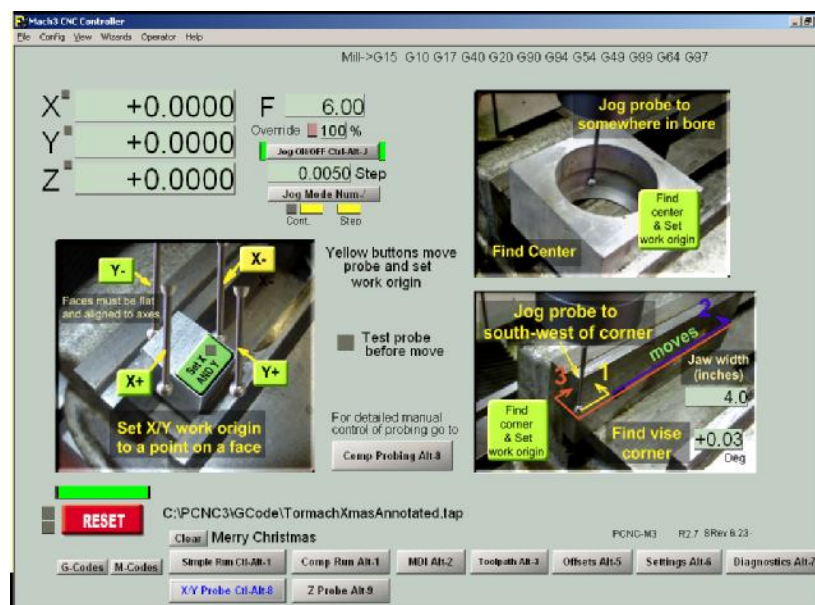


Figure 8.10 – Simple XY Probing

We strongly recommend use of the Jog/Shuttle controller for jogging the probe position. because it combines precision with speed. If you hit anything while jogging the machine does not stop and, although the probe tip has a lot of over-travel, there is a risk of bending it or breaking the ruby tip.

Bore Center

Jog to position the probe tip so it is just below the top surface of the bore to be probed and anywhere roughly near the center. No precision is needed in this position. Click the Find Center and Set work origin button.

The probe will make three pairs of exploratory moves. First it goes left and right (X – then X++) and so estimates the X coordinate of the center. Then it goes to the estimated X-center and probes back and forward (Y—and Y++). This gives the exact Y center. Finally, using this, it refines the X center by the final pair of moves (The initial X moves off the Y center could have been slightly inaccurate because of the oblique contact of probe tip with the bore).

Then the current work offsets (typically by default the G54 system) are set so the bore center is 0,0 (the X/Y origin). As the probe tip is in the center the X and Y axis DROs will read 0.0.

Position of a Face

You can set the X origin of a plane face perpendicular to X or the Y origin of a plane face perpendicular to Y. The Yellow buttons X+, X-, Y+ and Y- determine which direction the probe will move. You should start within ½" or so of the face and with Z low enough so the tip hits the face.

After the probe trips it moves slightly away from the face that it has found and the control software sets the corresponding current work offset so that the face is coordinate 0.0. This is typically useful to set the two edges of a piece of material to be the origin of their axes (i.e. the corner will be 0.0, 0.0)

There are occasions when you actually want to set both X and Y offset values to the location of the point probed. To do this toggle



Figure 8.11

the green button Set X AND Y before making the probe move (figure 8.11). Two different moves will not, of course, be meaningful if they use "Set X and Y" as the second overrides the first.

The probed face should not be significantly angled to the X or Y axes and should be flat. Any errors here will be reflected in the accuracy of the probed position. If you need to probe inclined and/or radiused faces then you need to look at the Comprehensive X/Y Probing screen.

Corner of Vise

It is very quick, for machining a small batch of parts that can be held in a vise, to align the stock by feel to the corner of the fixed jaw. This function allows you to find the back left corner which is a typical choice for X = 0.0 Y = 0.0.

Jog the probe to ¼" to ½" down and to the left (South-West) of the corner of the fixed jaw. The Z height make the probe tip be below the top of the jaws. Set the Jaw width DRO to the size of your vise (this will be remembered for subsequent uses of the screen)

The probe explores using the three moves shown on the screen panel (figure 8.12)

Move 1 establishes the rough position of the jaw. Move 2, near the far end establishes the angle the vise is mounted at (and displays it in the DRO). Move 3 approaches the end of the jaw (along a perpendicular). The X/Y origin of the current work offsets is then set to the calculated position of the corner and the probe moves to its starting point.

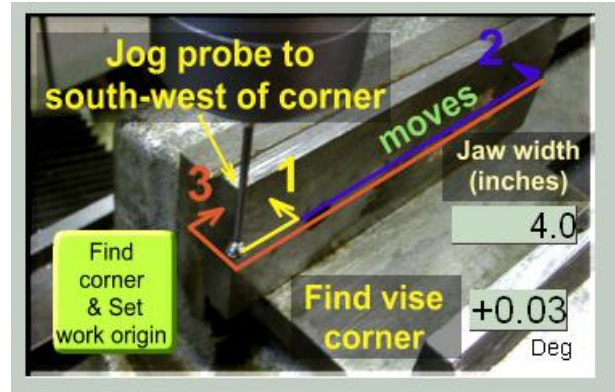


Figure 8.12 – Vise jaw corner

The angle DRO gives you a confidence check the setup accuracy of your vise. If it is a long way from 0.0 degrees then Move 2 might not find the jaw (or might even hit the moving jaw). If you need to work with a significantly skewed vise or to very close tolerances then the Comprehensive X/Y probing screen has features that will help.

Notes for all operations: You can use these operations with a part program loaded or indeed with one partly run (say to an optional stop). The probing will however set the Control Software to Absolute

Coordinates (G90),
No rotation (G69),
No axis scaling
(G50) and cancel
canned cycles
(G80)

8.2.2.2 Z Probing

Figure 8.13 shows the Z probing screen. It has two distinct functions depending on whether the probe is in the spindle or a normal tool is in the spindle and the tool setter is connected to the

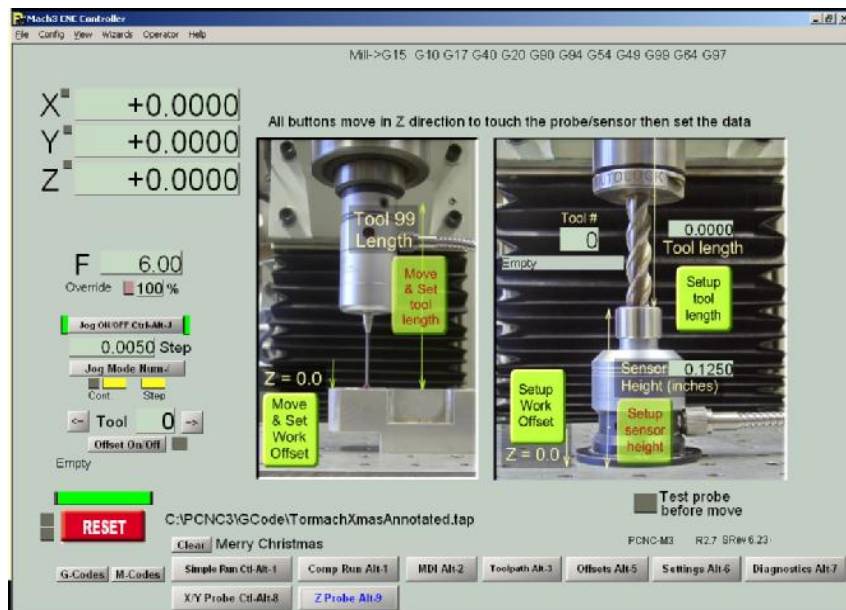


Figure 8.13 – Z Probing screen

probe input.

The yellow buttons perform machine moves and set offsets. Depending on the button, these will be the Z offset in the current work offsets or the length of a tool.

Important: If the probe or tool setter is not properly connected then the Control Software will keep moving even when the probe touches or the setter is touched. This can cause the probe tip to be crushed or the cutting edges of the probing tool to be chipped on the carbide top of the setter - the PCNC is very powerful! Each time that you connect up the probe or tool setter you should check that the screen LED marked Test probe before use lights when you gently displace the tip or cap. The red lights in the device will illuminate too but the screen LED is a more complete test.

We strongly recommend use of the Jog/Shuttle controller for jogging the probe position. because it combines precision with speed. If you hit anything while jogging, the machine does not stop and, although the devices have some over-travel, there is a risk of damage.

Probing a Z Surface

If the Control Software knows the length of the probe relative to the master tool then you can set the current work offset so any surface you probe is $Z = 0.0$; conversely if you have a $Z = 0.0$ surface then you can set the length of the probe in the tool table. The pane that does this is shown in figure 8.14.

We describe setting the tool length first although it needs to be done very infrequently (e.g. if the probe tip is changed or the master tool length has to be altered). The yellow button is semi-transparent to reflect the infrequent use and visually draw you to the Set Work Offset button.

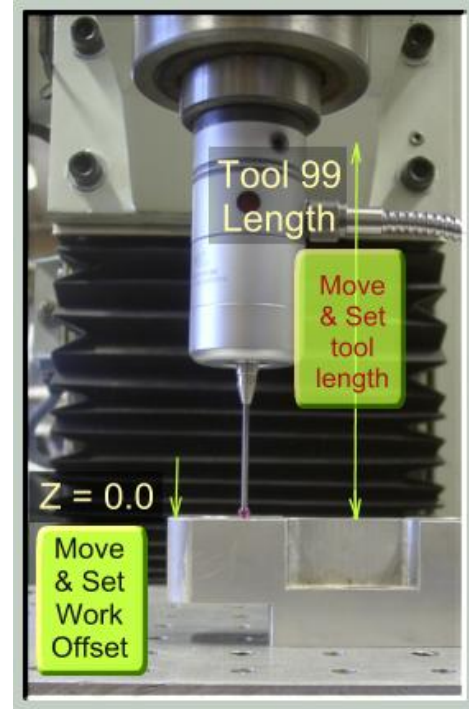


Figure 8.14 – Probing Z

- Put the master tool in the spindle and type its number (traditionally tool #0 or #1) into the T DRO. Jog Z so it just touches the table or any flat Z surface. Zero the Z axis DRO. You can use this or any of the standard setup processes for the Z work offset (for details, revise offsets and in particular sections 6.1 and 6.3)
- Load the probe into the spindle
- Click the button Move and Set Tool length so the probe trips on the same surface you used to touch the master tool. This will set the length of Tool #99 (the probe) relative to the master tool.

In everyday running you can now set any surface to be $Z = 0.0$. Just put the probe tip above the surface that is to be $Z = 0.0$ and click Move and Set Work Offset. This is the best way to set the Z work offset if the probe is in the spindle for setting X and Y too.

Setting a Tool or Z Surface

The other pane on the Z probing screen is used with the tool setter. It is shown in figure 8.15. Here there are three quantities and provided two are known the third can be quickly set. The height of the tool setter sensor is essentially a constant so needs to be put into the DRO once after each installation or upgrade of the Control Software.

We first describe setting the sensor height.

- Put the master tool in the spindle and put its number in the Tool # DRO

Machine upgrades and configuration

- Touch it on a flat Z surface like the table of tooling plate and set the current work offset so $Z = 0.0$ (sections 6.2 and 6.3 give a range of methods).
- Jog the master tool so it is above the sensor. Tests its wiring by depressing the top and checking that the Test Probe before move LED lights and click the Setup Sensor Height button. This will set the persistent DRO recording the sensor height.

To set tool lengths in the tool table you work as follows:

- Select the master tool. Touch a Z surface with the master tool and set work offset so $Z = 0.0$ as above.
- Put the number of the tool you are measuring into the Tool# DRO and put the tool in the spindle.
- Click the Setup Tool Length button. The tool will move and be sensed by the setter and its length relative to the master tool be entered in the tool table.

To set a plane as $Z = 0.0$ with a given tool

- Load the tool in the spindle and put its number in the Tool# DRO
- Place the Tool Setter under the tool on the surface to be made $Z = 0.0$
- Click the Setup Work Offset button. The tool will move down onto the setter and the work offset will be reset.

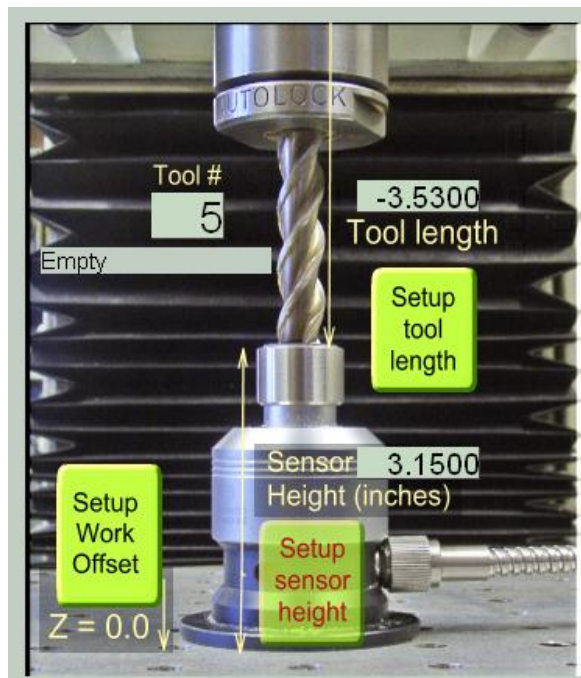


Figure 8.15 – Tool setter

Notes:

(a) If the setter is not directly under the tool then the probing move will miss the setter and so continue until the tool hits the table or fixture. If you see this happening trip the sensor by hand or use EStop to abort the move.

(b) You can use the Setup Work Offset button, and the setter, for tools held by a system like ER collets where you do not know the tool length. You must set up the setter height correctly but then click Setup Work Offset **whenever** you change a tool. In this case the tool length does not matter – everything is done by the Z work offset. You **must not** mix this method of working for some tools with using the tool table for others because of the differing use of the work offsets in the two methods.

8.2.2.3 Comprehensive X/Y Probing

The simple X/Y probing screen is designed so that the one click does everything needed to set the required offsets. This imposes the limitation that the moves are performed by the software rather than by the operator. For example while you can find the center of a circular bore you cannot find the center of a circular protruding boss even though the basic geometry is identical. A Comprehensive Probing screen is provided to overcome this limitation. The screen is accessed by a button on the simple X/Y probing screen (figure 8.16).

Figure 8.17 shows the Comp X/Y Probing screen.

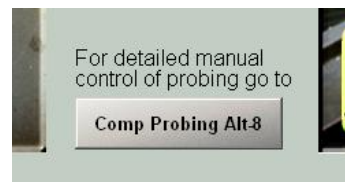


Figure 8.16 – Accessing the Comp Probing screen

It gives the operator the required flexibility by separating the actions of probing for feature on the work and setting work offsets from the features. The interface between the operations is the X/Y coordinates of three points which are stored and displayed in a bank of DROs.

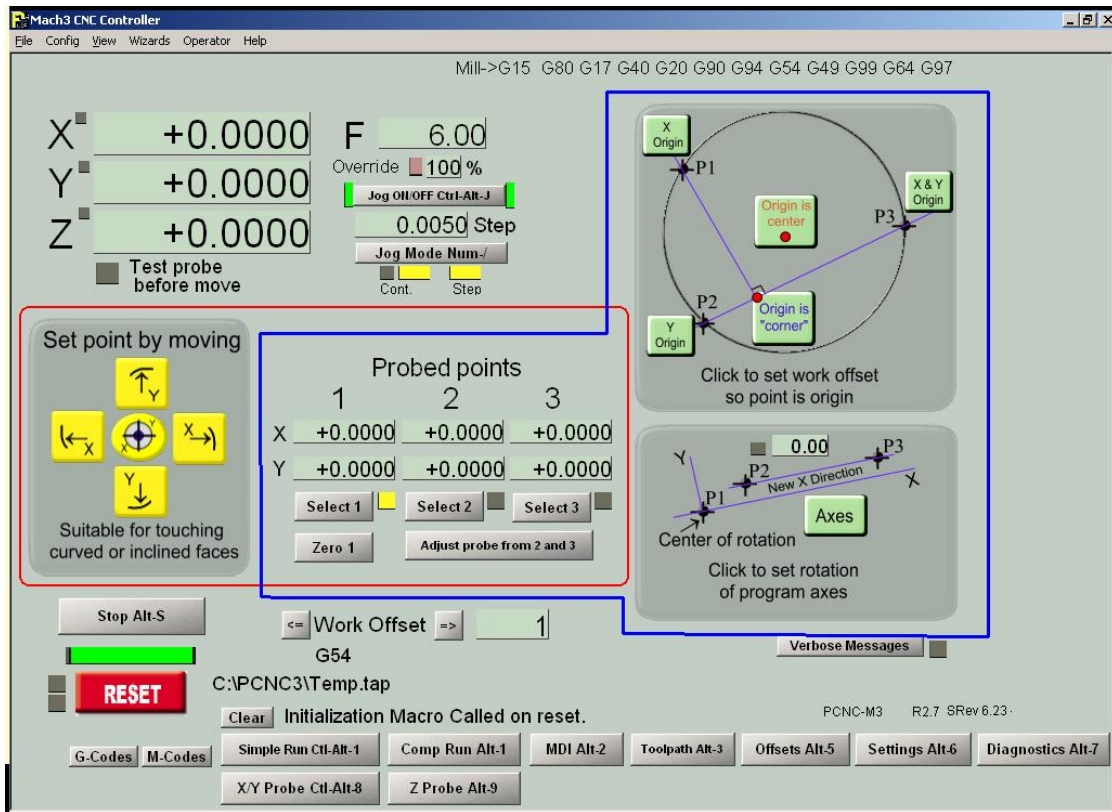


Figure 8.17 – The two areas of Comp X/Y Probe screen

The probing and recording function is done using the yellow buttons and the DROs enclosed in the red box of the figure. The offset setting is done with the DROs and green buttons enclosed in the blue box.

Probing to Set Point DROs

Before probing you should set which Point will be used to record the probed coordinates by clicking Select 1, Select 2 or Select 3. The adjacent LED will indicate the currently selected point.

The circular central yellow button will set the Point to the centre of the circular hole in which the probe tip is placed. Ensure that the Z axis is lowered sufficiently so contact is made on the "equator" of the tip.

The cross-pattern of square buttons probes in the two X directions and the two Y directions. Unlike on the simple X/Y Probing screen, the face being probed does not need to be square-on to an axis and can be a circular arc segment. This is indicated by the graphic on the buttons.

The angle and possibility of an arc are detected by making a main probe move followed by two side-steps 0.2" either side of the main move. The coordinate recorded in the selected Point DROs is the point where the main move contacts the work. The diameter of the probe tip is allowed for so, as for simple probing, it must be correctly entered for tool #99 in the tool table. The probe is returned to its initial point.

Progress of the operation is noted on screen in the message line. If Verbose Messages are selected (indicated by the LED next to the toggle button) then details of what the probe found will be given. If the face has a radius of curvature of greater than or equal to 20" the angle of the face (or tangent to the large circle it is part of) will be given. If the radius of curvature is less than 20" then an estimate of the radius and the center point will be displayed. Clearly these

will not be particularly accurate as they are derived from three points that are very close together but this information can be a useful check on the setting of the work.

When one point has been probed then another can be selected and the DROs set by probing another feature.

In some situations you may not need to probe to set the Point DROs. You can just type values into then (followed of course by *Enter*). Point 1 can also be zeroed by a button.

Figure 8.18 shows probing for three points to find the center of a feature that could not be located by the simple X/Y Probing screen.

Setting Offsets from Points

Once you have a point or some points you can use them to set work offsets by using the green buttons.



Figure 8.18 – Finding a tricky center

Origin X, Y, or X and Y from a point: The buttons outside the circle use just one point. If you only want to set the X offset use Point 1. If you want to set just the Y offset use Point 2. To set both X and Y use Point 3.

Origin X and Y are center of an arc: Set all three Points to lie on the circle or arc. Click the Origin is Center button. Although the picture on the screen implies the points are roughly equally spaced around the circle, and this will give the greatest accuracy, you can work with a small segment of arc drawn from a large circle whose center is outside the machining envelope by many feet. As the points can be centers of holes it can find the center of a set of holes on a pitch circle like a pipe flange.

Origin X and Y are at intersection of line and perpendicular from a point: Again you set all three Points. Points 2 and 3 define the line and Point 1 from where the perpendicular is to be dropped. Click the Origin is "corner" button.

The picture implies that the perpendicular should intersect the line between the probed points but this is not necessary. The "corner" can be on any projection of the line. This function is a generalization of the vise jaw locator on the simple screen. It can set the origin to any corner of an object in any orientation.

Setting Rotation from Points

This function is used to set up the machine to mill components where the X/Y axis of the part is not square to the X/Y axis of the machine. This might be because the part does not have any square faces accessible when it is mounted for a second operation. The function uses the G68 code to rotate the working coordinate system relative to the machine coordinate system. You should be sure you understand the working of G68 by running test pieces before you use it on a job.

Point 1 defines the center of rotation in the existing work coordinates. It is often convenient to keep this as 0,0 or for it to be the same as one of the other points. Points 2 and 3 define a line whose angle is the direction of the new coordinate system.

If no flat face is accessible it will often be useful to have Points 2 and 3 as the centers of holes already drilled or bored in the part.



Figure 8.19 Probing Point 2 – Cable right

Click the button Axes to set the G68 command. The angle of rotation used is displayed in the DRO and if it is non-zero the LED on this screen and next to all axis DROs will indicate that G68 is active. The X and Y DROs will be red to remind you the rotation is active.

Rotation is removed by clicking the Clear Rot button, by typing 0 into the angle DRO or by using a G69 command.

8.2.2.4 Probe Calibration

For the probe readings to be meaningful the center of the probing tip ball needs to lie on the centerline of the spindle. The Tormach probe has three set screws which are used to adjust this alignment. Most other probes will have similar features although the pitch of the threads and the orientation of the screws may differ and so require alteration to the details of this procedure.

To align the probe tip you need a hole about ½" diameter and ¾" or so deep machined in a block of material so that it is perpendicular to the bottom face of the block.



Figure 8.20 – Point 3 – Cable left

The hole should be as circular as possible so ideally be reamed or bored on a lathe. If it is produced on a lathe then having a through hole and facing the end provides the accurate bottom face. A rough setting can be made using the TTS Measurement Fixture (p/n 30140) although its bore is not finish machined.

- Insert the probe in the machine spindle (in the collet with which it will be used) so that the connector and cable are on the right (X++) direction. Leave the spindle lock on.
- Work using the Comp X/Y Probing screen. Zero the Point 1 DROs as it is not used. Select Point 2 and use the round yellow centre finding button to record the center of the alignment bore. This is illustrated in figure 8.19.
- Rotate the spindle by half a turn and re-lock it. The cable will now be to the left (figure 8.20)
- Select Point 3. Again use the round yellow button to find the center of the alignment bore. If the probe was perfectly set points 2 and 3 would be coincident. They probably will not be so you need to know how to adjust the screws.
- Rotate the probe back to the Cable Right position.
- Click the Adjust Probe from 2 and 3 button. The message line will display the movement required on the three screws to centralize the probe tip. The screw opposite the connector and cable is Screw 1 and the others number clockwise looking from the top. So Screw 2 is facing the control cabinet and Screw 3 is visible on the "southeast" side of the probe. The numbers are included on the probing photographs above for additional clarity. A positive number is tightening the

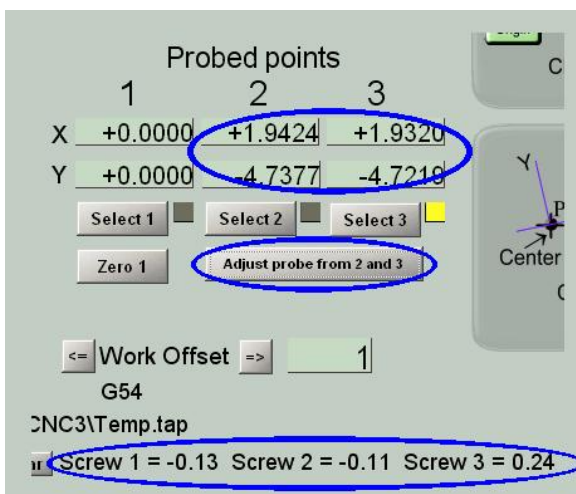


Figure 8.21 – Controls for probe alignment

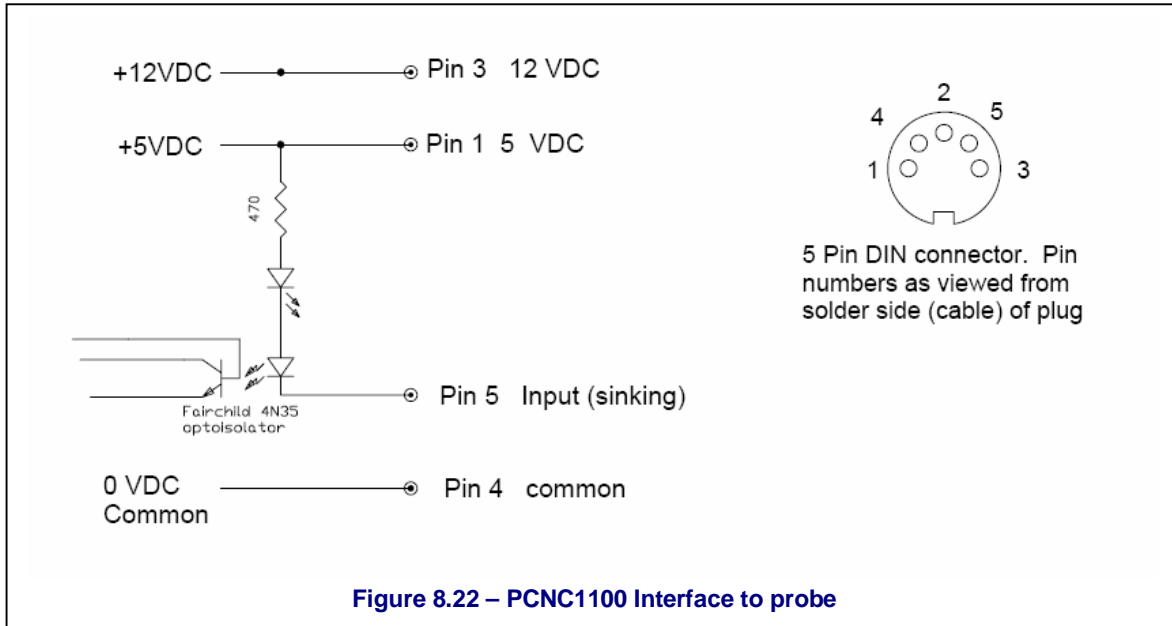
screw and a negative number loosening it. The value is millimeters which equates to turns on the Tormach probe. You must loosen the screw(s) first then tighten the other(s).

- You will not be able to estimate, say, 0.11 turns so get as near as you can and repeat the procedure getting nearer and nearer to the ideal point where all screw movements are zero. For most purposes 0.03 of a turn which is about 0.001" will be good enough.
- Make sure all the screws are tight and re-check the calibration.

8.2.3 Digitizing parts from a model or for reverse engineering

Details of the software for this will be published in a later revision on this document.

8.2.4 The Probe Electrical Interface



Probes are connected to the auxiliary socket is provided on the operators control panel. This can be used for interfacing any device that needs an input that can be read by the Control Software. It can be used for user supplied devices. The exact interfacing will depend on the device to be used but the following description of the available signals will allow liaison with the device supplier/designer.

The interface is presented as a 5 pin DIN socket. The pin assignment is:

Pin number 1 is +5 Vdc, useful if you have to bring power to active electronics on a probe.

Pin number 3 is +12 Vdc, useful if you need to supply power to active electronics on a probe.

Pin number 4 is logic ground. Do not attach to the machine ground (i.e., your probe casing should be isolated from this ground).

Pin number 5 is the signal. It is internally pulled up to 5 volts in the PCNC. The input is active ("on") when this is pulled to ground. Thus by jumpering 4 to 5 and you will see the probe LED change on the Diagnostics screen.

8.3 Auto-reverse tapping

The control software supports auto-reverse tapping heads supplied as accessories to the PCNC. PN 30612 ranges from #0 to 1/4" threads and PN 30613 covers #8 to 1/2". The DROs on the Settings screen are used to define the head in use and thread pitch. The actual threading commands M871, M872, M873 and M874 are described in chapter 7.

This section gives brief instructions of fitting the heads to the machine and installing a tap.

Both tapping heads are supplied with the Tormach Tooling System geometry whose 3/4" shank can be mounted like any other tool.

Figure 8.23 shows the large head fitted in the spindle. The torque arm can be seen engaged in the recess in the head casting to prevent rotation on the fixed part of the mechanism. Tapping will generally be performed between 500 and 900 RPM so the drive should be set to the low speed pulleys.



Figure 8.23 – Large tapping head ready for use

The tap is held by a rubberflex collet locked by the nut on the nose of the head spindle and prevented from rotating in the head by using its squared end. The small head accepts a fixed square but the larger head, which covers a bigger range of sizes, has a set of alternative blocks which fit into a square hole above the collet nut. The tap holding arrangement, without the spindle is shown in figure 8.24 and the accessories for the large head are shown in figure 8.25. This picture also shows the square hole in the tapping spindle which houses the anti-rotation block.

The top ring on the tapping head should be screwed down to give the highest tapping torque. As the machine is applying the feed any rotational slippage will cause problems.

Tapping can be done using the M871 (M872 etc.) cycles, tapping code generated by CAM systems such as SheetCAM and, of course, hand written G-code. The recommended sequence is the same in each case (figure 8.26):

- Feed the Z axis down at a rate corresponding to the pitch of the thread and the spindle speed. The distance of this move will be the depth to be tapped plus the initial Z clearance.
- Rapidly retract the tapping head. As the tap is held in the thread this will pull the spindle out of the head and engage reverse gear.

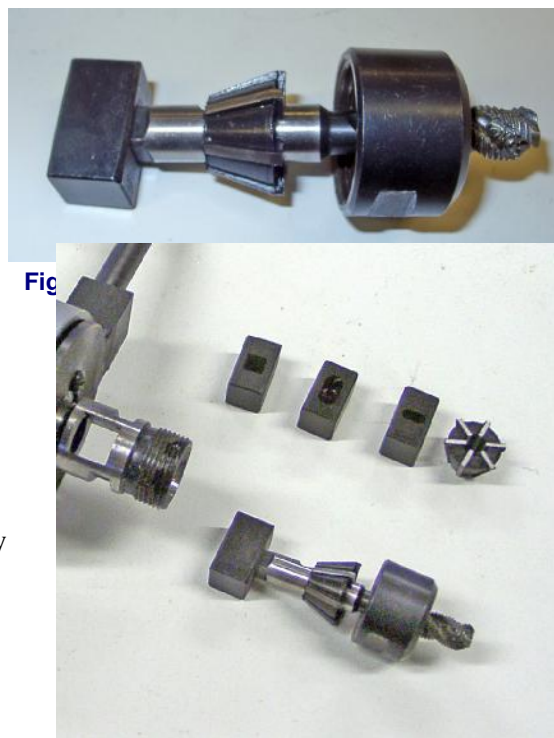


Figure 8.25 – Accessories for large tapping head

Machine upgrades and configuration

- Feed the Z axis up at a rate corresponding to the pitch of the thread and the, geared up, speed of the reverse running tap. The distance of this move should be sufficient to ensure the tap completely leaves the thread.
- Optionally position the Z axis at the initial Z clearance height.

Tapping with M871-M874 Macros

The M871 – M874 tapping macros use 4 Z moves:

1. Feed down at the tapping feed rate to tapping depth.
2. Rapid retract to reverse the tapping head.
3. Feed up at the tapping reverse feed rate to clear the tap.
4. Rapid down to the starting point.

Call using one parameter: M871 Px.xxx (Px.xxx = Z start – Z final)
See sample procedure below.

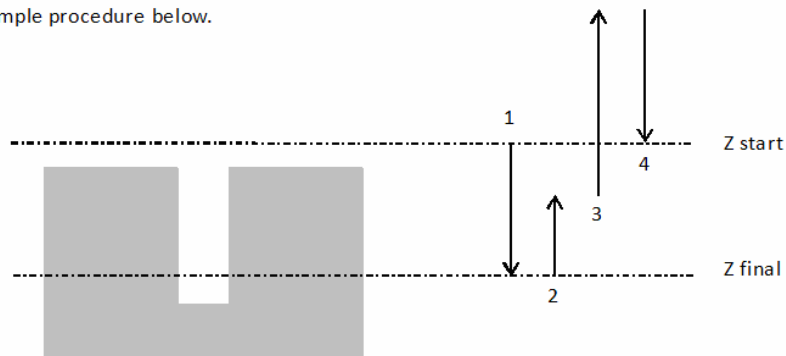


Figure 8.26 – Moves in tapping a hole

Notes:

It is usual to underfeed slightly in both directions (2% to 5%). This might result in the clutch disengaging but it is better than hitting the stops on the retractable spindle.

The choice of a suitable tap designed for machine tapping is important. Conventional taps for manual use are unlikely to give satisfactory results except on shallow depths with easy to machine materials.

The exact depth fed depends on the accuracy and stability of the spindle speed. This makes tapping blind holes where the thread depth is nearly as deep as the drilled hole, a process which requires experiment to get the optimum depth. There are alternative feed strategies which can increase the accuracy of tapping depth with the trade-off of greater wear on the tapping head clutch.

In tapping blind holes, it is common practice to rough tap by machine and then clean the hole and the bottom the thread with a hand-held tap or pneumatic power head.

If you are writing your own G-code, the rapid retraction is best done by a G01 move with a large feedrate (say F99999) as this ensures that the control software Constant Velocity is used to make the fastest and smoothest reversal. CV would not be used if G00 is employed.

During the start of the rapid retraction the tap is still actually moving into the hole then stationary. This should be allowed for when



Figure 8.27 – Measuring spindle speed

calculating distances.

For best results, especially with long threads, you should check that the actual spindle speed corresponds to that commanded by the S word by using a tachometer (e.g. Tormach p/n 30527) (figure 8.27)

9. Warranty, Specifications and Maintenance

9.1 Intended Use Statement

The PCNC 1100 is intended for use as a general purpose CNC milling machine. The intended use includes cutting conventional (non-abrasive) materials such as unhardened steel, aluminum, plastics, wood and similar materials. The PCNC 1100 is intended to be used with the software configuration files provided by Tormach.

9.2 Support

Tormach provides free technical support through multiple channels. The methods are listed below, in order of preference. The quickest way to get the answers you need is normally checking in order of preference:

- **This manual – ALWAYS the first place to check!!**
- **Related documents found at:** <http://www.tormach.com/documents.htm>
- **Our website at:** www.tormach.com
- **Email to:** info@tormach.com
- **Telephone Tormach at:** 608-849-8381
- **Fax Tormach at:** 209-885-4534

9.3 Outside of the Scope of Intended Use

Applications for the equipment or modifications of the equipment outside of the Intended Use Statement are supported through consulting engineering, not through our free support policy. There are no limits to the applications that Tormach products can be used for or to the modifications that can be applied to the Tormach machinery. Tormach designs use standard industrial components and incorporate the principles of Open Architecture specifically to allow and promote these variations. With Open Architecture controls, industrial engineers will find Tormach products cost effective to incorporate into larger manufacturing systems, or with easily separable base, column and head sections manufacturing engineers looking to design specialized in-house equipment will find they can use the base as a low cost motorized XY table. Some machinists may want to convert a stepper mill to servos or some software engineers may want to replace the Tormach approved software with something of their own creation.

All of the technical information and insight required to support these variations from the intended use cannot possibly be foreseen. If the extensive documentation provided does not supply all the information you need, we can provide additional information and engineering support required for your project on a consulting engineering basis. We are actually very good at this. If you have your questions well organized we can normally provide all the information you need in short order. Consulting engineering is done by electrical and mechanical engineers and billed at current hourly rates.

As you might expect, all warranties for Tormach equipment are voided through modification to the equipment or use outside of the Intended Use. Individuals or companies involved with modifying the equipment or applying the products assume all consequent liability.

9.4 Specifications

9.4.1 Mechanical

- Table Size – 34" x 9.5"
- Table Slots – 3 T-Slots. 5/8", 2 Drainage/Alignment slots
- Travel (X-, Y-, Z-axis) – 18" x 9.5" x 16.5"
- Spindle Nose Diameter – 3 ³/₈"
- Spindle Taper – R8 with spindle lock for easy single hand drawbar operation.
- Spindle Speed Range RPM – 300-4500. Computer controlled spindle speed and direction with DC injection braking from a 1.5 hp induction motor (2 hp peak).
- Cartridge style spindle with V-belt transmission. There are two overlapping speed ranges. Low belt speed is 300 – 1750 RPM, high belt is 800 – 4500 RPM
- Spindle Center to Column Face – 11"
- Maximum Weight of Workpiece – 500 lbs.
- Max spindle nose to table – 17"
- Forward opening electrical cabinet. This unique design allows service even when the machine is placed near to a wall.
- Weight net / shipping. – 1130 / 1325 lbs
- Rapid traverse X, Y, Z – 65 IPM
- NEMA 34 stepper motors on X, Y and Z
- Single Shot Lube Pump
- 13 point lubrication. Each axis has an oil distribution channel milled full length on left and right vertical and horizontal surfaces (all 4 sliding surfaces) plus an oil line plumbed directly to the ball nut assembly.
- Precision ground ball screws in the X, Y and Z motion. ISO/DIN P4 grade.
- Complete way covers.
- Slide ways are a bonded low friction surface (PTFE filled acetyl), hand scraped slide ways with milled distribution slots.
- 6 month warranty. There is no on-site factory service personnel. Warranty service will involve replacement of components with direction from Tormach support staff.

9.4.2 Electrical

- Spindle safety is enhanced with a tool change safety interlock and key switch lockout. Main spindle contactor (VFD power) is automatically disengaged when operator is changing a tool. Spindle M-codes or manual spindle start will reengage VFD power automatically after tool change is completed.
- Operator console has manual override for all spindle controls.
- PC Windows Control with integrated safety & manual overrides (PC not included, Windows XP is required). Computer control is integrated with an operator control panel for spindle lockout, manual spindle controls, manual coolant control and PC power control.
- Digitizing & tool touch off port, optically isolated.

- Coolant or accessory AC outlet. The AC outlet for coolant is automatically controlled via M-codes while the operator console offers a convenient manual override.
- PC or accessory AC outlets, manually controlled on the operator console.
- Power requirements – 230 VAC single phase (110 VAC option)

9.4.3 **System**

- Complete RS-274 (G- & M-codes) implementation with unlimited program size. Conventional G- & M-code operations are complemented with manual mode operations, operation wizards, DXF drawing support, HPGL, bitmap and JPEG toolpath generation. Operation Wizards allow pocketing, surfacing, cutouts, drill patterns and other standard operations to be completed without the need for addition software or manual code writing.
- Supported codes include: G00 [Default] Positioning (Rapid Traverse), G01 Linear Interpolation, G02 Circular/Helical Interpolation CW, G03 Circular/Helical Interpolation CCW, G04 Dwell, G17 [Default] Circular Move XY Plane, G18 Circular Move ZX Plane, G19 Circular Move YZ Plane, G20 [Default] 06 Inch Input, G21 Millimeter Input, G40 [Default] Cancel Cutter Compensation, G41 Cutter Compensation Left, G42 Cutter Compensation Right, G53 Work Offset Cancel, G54-G59 Work Offset, G74 Incremental Input I, J, K, G75 [Default] Absolute Input I, J, K, G80 [Default] Canned Cycle Cancel, G81 Spotting Drill Cycle, G82 Speed Peck Drill Cycle, G83 Full Retract Peck Drill Cycle, G85 Boring Cycle, G86 Boring Cycle – Spindle Stop, G87-89 User Definable, G90 [Default] Absolute Input, G91 Incremental Input, G92 00 Set Program Part Zero, M00 Program Stop, M01 Optional Program Stop, M02 Program End, M03 Spindle Clockwise, M04 Spindle Counter-clockwise, M05 Spindle stop, M06 Tool change, M07/M08 Coolant, M09 All Coolant Off.

9.4.4 **Options**

The PCNC 1100 comes preconfigured for low cost 4th axis addition. Software support, wiring, cabinet holes and drive mount holes are all in place for an easy 4th axis addition.

A wide range of accessories such as tapping heads, probes, tool setting sensors are available.

9.5 **Maintenance**

9.5.1 **Protecting from Rust**

Exposed iron and steel surfaces should always be protected from rust and corrosive environments. If your machine will be unused for more than a couple days you should mist the machine with light water repellent oil such as WD-40.

The rust preventative characteristics of coolant may not be effective in trapped areas. It is desirable to apply way oil or machine oil directly to the table surface under the trapped area when you mount a vise or fixture on the machine. Neglect this and you may be welcomed by a rusty machine surface when you remove the fixture some days later.

9.5.2 **Lubrication**

The central lubrication pump should be filled with quality way oil. This could be Tonna 68 (Shell), Vactra No. 2 (Mobil), Way-lube 68 (Sunoco), WayLube 68 (Texaco), Febis 68 (Esso) or equivalent oil.

A shot of lubrication should be given for each 4 hours of operation and after the machine has stood unused for more than 48 hours.

Always make certain that the lubrication oil is clean. The oil is delivered to 15 points throughout the machine. This includes the 12 sliding surfaces (4 each on the 3 axes) and 3 ballscrew nuts. These are some of the most critical and expensive mechanical parts of the machine. Any dirt or foreign material suspended in the oil is going to be delivered directly to these parts and can dramatically shorten the effective life of the machine.

Be sure to clean off the cover and surrounding area before refilling the oil reservoir. The strainer at the top of the reservoir is only a screen; it is not a filter.

Note:

- The pump is spring loaded, where the spring force creates a very light hydraulic pressure. You can get the oil out quicker by pushing a bit, but apply too much force and you can pop off one of the oil lines.
- You will have a more uniform distribution of oil if the machine is moving when the hydraulic pressure is applied.
- The pump sucks up oil from the reservoir on the pull stroke and delivers it to the machine on the push stroke. If at some point the oil pump seems much easier on the push stroke then make certain that you do not have a broken oil line.
- Extreme axis positions can expose the oil distribution channels that are cut into the way support saddle surfaces. If the pump is used in those positions the hydraulic force of the oil will not apply it throughout the machine as intended. Instead the oil will simply squirt out at the point where the oil channel is exposed.

9.5.3 Way Covers

Way covers are important to keep abrasive debris out of the slide ways. Inspect the way covers frequently and replace as necessary. Tormach stocks way cover replacements.

9.5.4 Z-axis Adjustment

When the power is turned off the spindle head is held up against gravity by a combination of gib pressure in the vertical slideway and the cogging force of the stepper motor. If the spindle head moves down on its own when the power is off, then it is time to adjust the Z-axis gib. This is common after the initial break-in period of the machine.



The slide ways have tapered gib plates, where the position of the gib plate controls the tightness and friction in a slide way. On the exploded view (Appendix 2), the Z-axis gib plate is part number 80. This is held in position on both top and bottom via the gib screws, part number 81. If the Z-axis drifts down when the machine is turned off, then the Z-axis gib plate needs adjustment.

Adjustment Procedure:

CAUTION: Do not remove the connector (exploded view #71) that joins the Z-axis motor to the Z-axis ballscrew. Do not electrically disconnect the motor from the motor driver, nor disconnect the motor driver from the DC power supply. Any of those actions can allow the spindle head to lower with a dangerous and damaging velocity.

The gib plate is tapered down. When the gib plate is moved down relative to the Z-axis saddle (part 82) the slide way will become tighter, when it moves up the slide way will become looser. With the machine is turned on, move the Z-axis up to a convenient height. Remove the screws that attach the way cover (part 74) to the Z-axis saddle (part 82) in order to gain access to the lower gib screw. Loosen the lower gib screw, backing the screw out about 2 or 3 turns. Then place a wooden block under the head to prevent it from hitting the table. Turn power off on the mill.

With power off and the head protected against hitting the table, adjust the upper gib screw in until it takes about 40 to 80 lbs of force to get the head started moving down. You can get it moving by pulling down on the head. The motion should stop quickly after you stop pulling down. If a 5 lb pull gets it moving, or if it does not stop moving down on its own after you stop pulling, then the gib is not tight enough. If you cannot get it to move down, either it is too tight or else you forgot to turn off the machine. You will not be able to move the Z-axis head while the machine is on. Once you have the gib adjusted correctly, tighten up the lower gib screw to lock the gib plate in place and replace the Z-axis protective cover.

Note:

The gib plate is tapered down. It can wedge in place, jamming the motion if the lower gib screw is left loose when the head is moved up. Always have the lower gib screw tightened when the head is moved up.

The head will never accelerate to a high speed moving down, as long as the axis motor is attached and the wires remain connected to the axis driver in the cabinet. This occurs because the motor acts as a generator during downward motion, taking energy from the system.

It is interesting to note that the LED on the Z-axis driver (inside the control cabinet) will glow dimly if the head drifts down under gravity with the machine off. This is due to the electrical generation of the Z-axis motor. **Warning:** The electrical energy generated is absorbed in the DC bus resistor. If the Z-axis driver were to be disconnected from the DC power supply then the LED on the Z-axis driver will glow brightly until the driver goes over voltage and self destructs.

9.5.5 Adjusting Ballscrew Thrust Bearing Preload

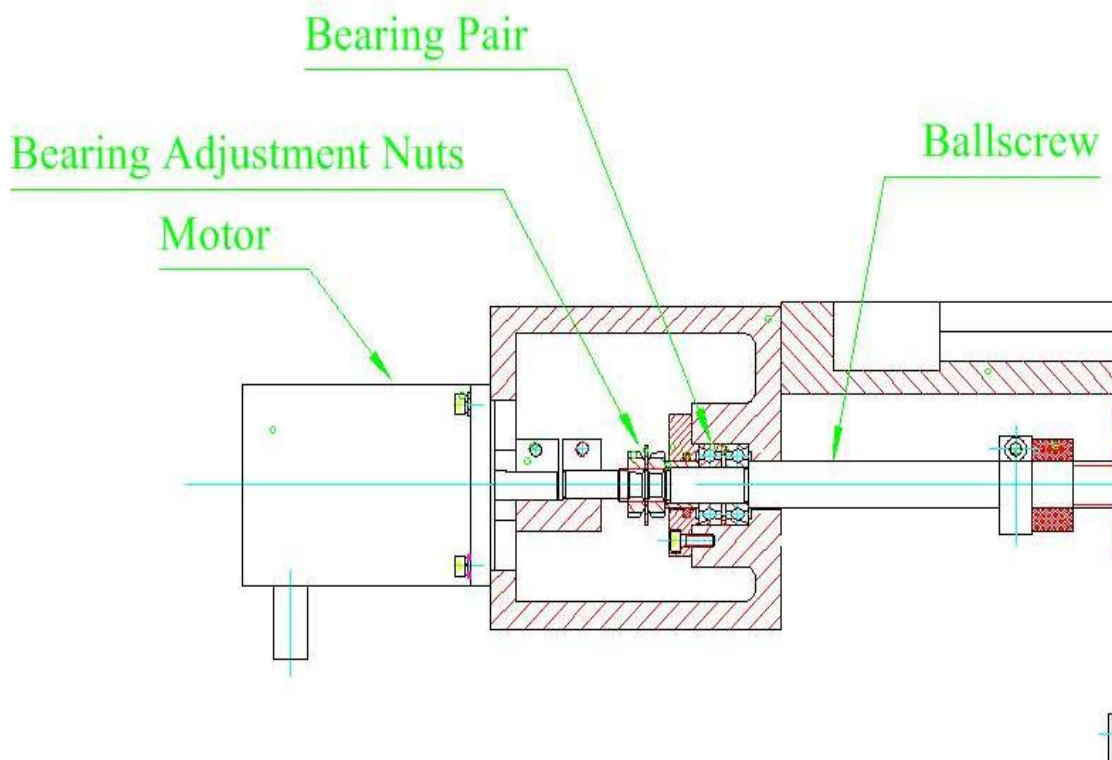


Figure 9.1 – Section view of X-axis mount

If the Bearing Adjustment Nuts are not set properly then there will be either excessive backlash in the machine (too loose) or rapid wear and excessive friction (too tight). Figure 9.1 shows the X-axis bearings, ballscrew and motor. The Y-axis and Z-axis are similar.

The recommended procedure for checking backlash is to observe motion at the table or head (via a dial indicator) while moving the machine using the axis motors. Tormach recommends adjusting the backlash on X and Y to something between 0.0004" and 0.0008". The recommended backlash for Z-axis is 0.0006" to 0.001".

Checking backlash by pushing or pulling on the axis is a less accurate method. The push/pull method will normally show a smaller backlash value. You can fool yourself into thinking there is zero backlash when the effective lost motion is within specification.

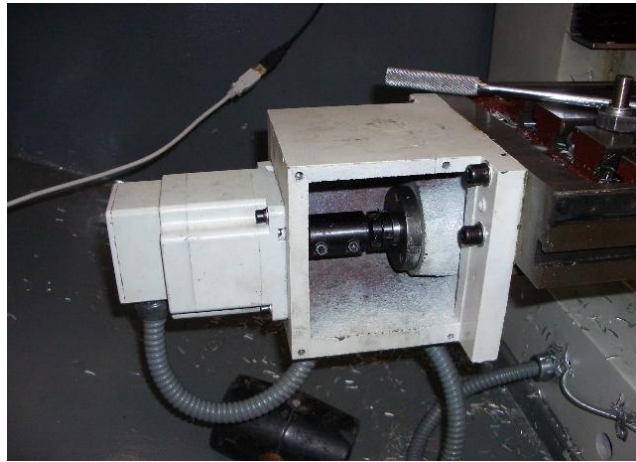


Figure 9.2 – Photograph of X-axis mount

9.5.5.1 Understanding Preloaded Angular Contact Bearings

Figure 9.3 shows a cross section of a typical ball screw shaft mount. The ball screw shaft is in the center and the crosshatched section is the iron casting that mounts the bearings. There are two angular contact ball bearings, forming a preloaded pair. The cover plate holds the two outer

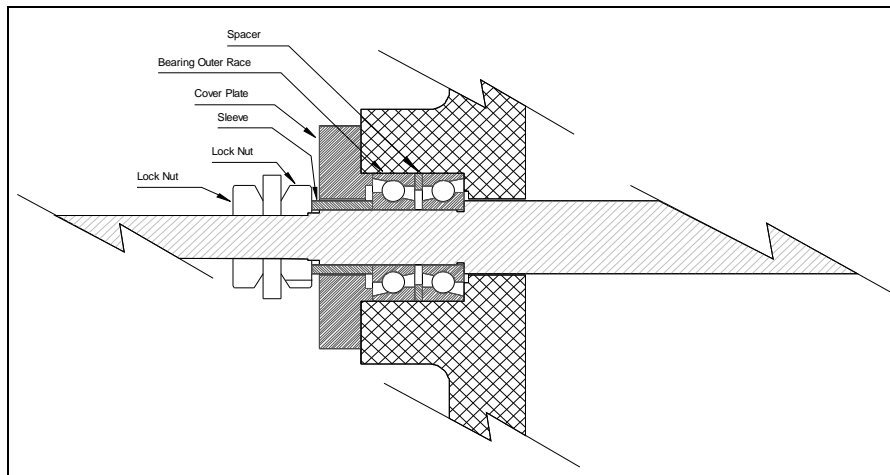


Figure 9.3 – Detail of bearing system

rices together, along with the spacer that is between them.

The inner races are held between the sleeve (left side) and the shoulder cut into the ball screw shaft. The sleeve is held against the left inner bearing race by the locknuts; as the locknuts are screwed toward the bearing pair, the preload increases.

Figure 9.4 shows how the force of preload is transmitted through the bearings, from the inner race to the outer race. In a preload pair, this force is then transmitted back to the inner race by an opposed bearing. It should be apparent that the correct orientation of the angular contact bearing is critical to the operation.

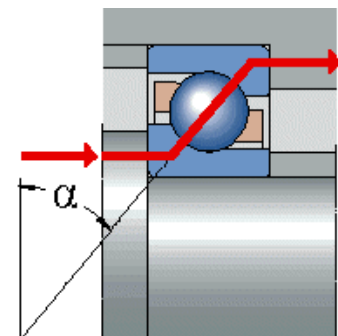


Figure 9.4 – Angular contact bearing

9.5.5.2 Making the Adjustment

In order to make the adjustment you will need two hook spanner wrenches (PN 30485) in order to rotate the two nuts on the ballscrew end (figure 9.5).

The nut nearer the bearing housing is the *adjustment nut* and the one nearer the stepper motor is the *lock nut*.

When working on the Z-axis you should remove any tooling from the spindle and support the head by resting the spindle nose on a block of lumber supported on a small jack.

Loosen the lock nut and back it off about two turns. Hold the ballscrew to prevent it from rotating and tighten the adjustment nut until there is slightly more backlash than you ultimately want to achieve – tightening the lock nut will slightly increase the bearing preload.

There are two possible methods to judge the preload:

- (a) You can use a dial test indicator to measure backlash between the ballscrew end and the structure to which the bearings are mounted (table for X-axis or machine frame for Y- and Z-axes);

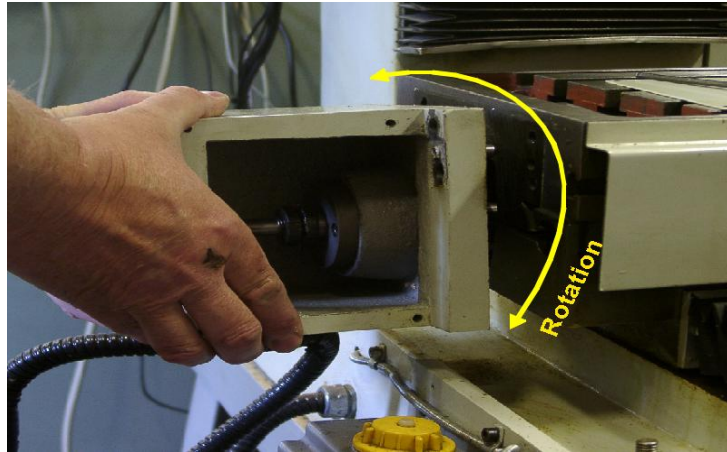


Figure 9.6 – Feeling preload in bearings

- (b) You can estimate the torque required to turn the ballscrew in the bearings.

If you are aiming for the smallest possible backlash then we suggest that you do both as it is not possible to be certain that you do not have **excessive** preload by just measuring backlash.

To estimate bearing drag torque you need to eliminate the stepper detent torque and the ballnut friction. This is done by allowing the stepper mounting box to rotate and measuring the torque required to do this.

- (a) For adjusting the X-axis position the table near the right hand end of its travel (i.e., X near to zero). This ensures that the bearing is near to the ballnut to minimize bending of the screw during tests.
- (b) Slacken the two screws clamping the coupling between the stepper



Figure 9.7 – Tightening the lock nut



Figure 9.5 – Spanner wrenches

shaft and ballscrew end.

- (c) Remove the four cap screws holding the stepper motor to the coupler box. Remove the stepper. This is the reverse of the process for coupling the Y-axis drive shown in section 2.3.
- (d) Remove the four cap screws holding the motor coupler box to the table (X-axis) or frame of the machine (Y- and Z-axis). Move the axis so the mounting box comes free of its pins. The screw and box are supported on the ballnut so take care not to apply forces that could bend the screw (figure 9.6).
- (e) You can now rotate the box by hand and get a sensitive feel for the torque caused by the preload on the bearings. Rotation should be smooth with a small perceptible drag this corresponds to a medium preload of about 150 lbs. If the rotation feels tight you have too much preload and will dramatically shorten the life of the bearings. If the rotation is free and/or the box can be rocked perpendicular to the screw axis then you have little or no preload and backlash will be excessive. This test should be done with the lock nut tight. It is best to support the box on its dowel pins when adjusting the nuts to avoid damaging the ballnut or risking bending the screw.
- (f) Finally remount the box and motor ensuring that the coupling is symmetrically fitted to the motor shaft and the screw end and is fully tightened (figure 9.7 & 9.8).

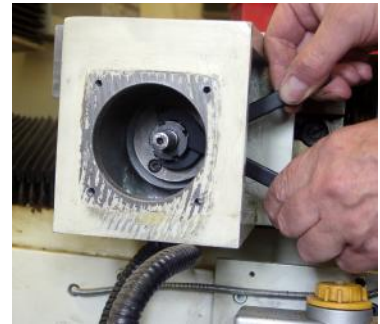


Figure 9.8 – Another view of tightening the lock nut

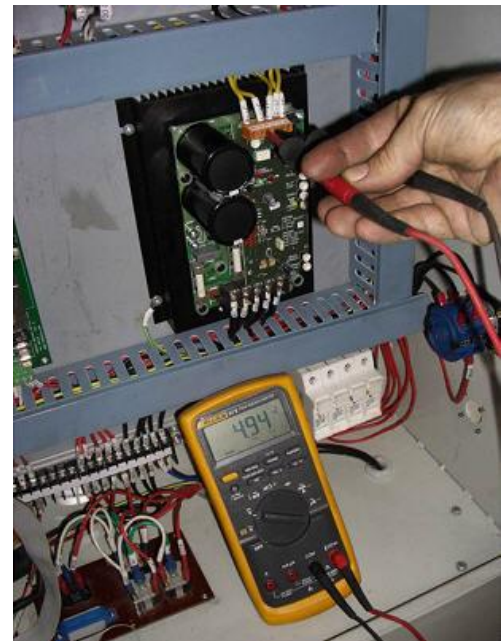


Figure 9.9 -Test Point Location

9.5.6 Speed Calibration

This procedure is not necessary for operation of a machine, it is only recommended as an improvement to spindle speed accuracy. The spindle speed in your PCNC 1100 is factory calibrated, but your computer is not calibrated to your machine. The variations among PCs will affect the precision of the computer generated spindle speed. To improve spindle speed accuracy, your computer needs to be calibrated to your PCNC 1100.

The PCNC 1100 uses a printer port of a standard PC for its machine control. The procedure described below will calibrate the PCNC 1100 to a specific computer. The procedure should be repeated any time the machine control computer is replaced.

While the voltage test points accessed in this procedure are limited to low voltage signals, the procedure does require access to the electrical cabinet while under power. Other points within

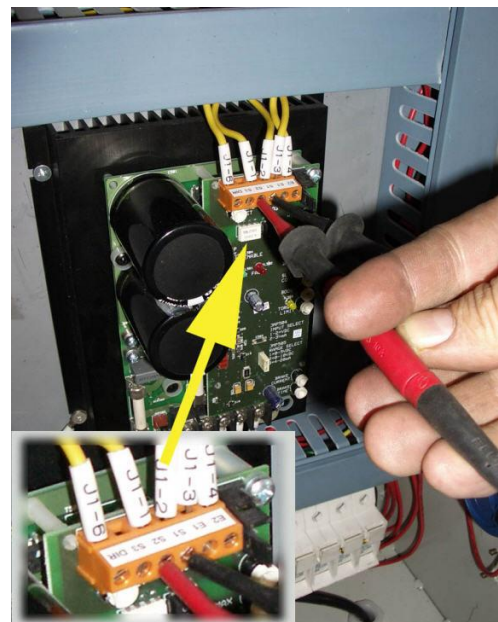


Figure 9.10 – Detailed View of Test Points

the cabinet will contain high voltage. Exercise caution whenever doing electrical tests. Only qualified electrical technicians should perform this calibration.

1. Turn on the machine and control computer and then open the PCNC control software program.
2. Turn the operator console **Spindle Lockout** key-switch to the OFF (0) position. This will prevent the spindle from starting. It also isolates the spindle motor driver from high voltage.
3. Turn the spindle control mode to **MANUAL** on the operator console. Turn the manual spindle speed knob fully clockwise, to the highest possible speed. Do not start the spindle.
4. Using a digital voltmeter set to measure DC voltage; measure the voltage from S1 to S2 on the spindle motor driver (figures 9.9 & 9.10). Connection point S1 is the same as wire number J1-2. Connection point S2 is the same as wire number J1-1. The black probe goes on S1 and the red probe goes on S2. Press the “Spindle Start” button on your machine. Record the voltage; it should be near 5V. This is your Volts on MANUAL measurement.

5. Now return to the operator panel on the cabinet door. Switch the spindle mode to **Auto**. On the PCNC control software, switch the belt speed position to HIGH and set the speed DRO to 4200 RPM. Turn on the spindle by clicking on the screen button marked SPINDLE. In step 3 you were creating the maximum spindle speed command from the operator console. Here you are creating the maximum spindle speed command from the computer. The actual spindle should not be turning because you should still have the spindle key lockout in the off position from step 2. Again, record the voltage; this is your Volts on AUTO measurement. Close the PCNC control software program.



Figure 9.11 – Opening the profile

6. Open the XML Tweak software program located under “Programs” on your “Start” menu. Click the “Open Profile” button and choose the machine profile you wish to modify (figure 9.11). The profile for the PCNC3 is C:\PCNC3\PCNC-M3.xml.
7. Next, under the “Speed Calibration” heading enter both of the measurements you recorded earlier. You will enter the Volts on MANUAL measurement in the *first* box. You will enter the Volts on AUTO measurement in the *second* box (figure 9.12).
8. At this point, you should click the “Save Profile” button. If you are sure all the information is correct, click the “Done” button. Close the XML Tweak software program.
9. Open the PCNC control software program again and confirm that the computer is calibrated with the machine by repeating step 5 from above. If the voltage falls between **4.57V – 4.77V** then it is within tolerance and is calibrated correctly. If not, start this procedure over.

Note:

Manually adjusting trimmers on your PCNC 1100 will put your speed reference out of calibration.

Manual adjustments can also affect speed stability and starting reliability. The XML Tweak program electronically adjusts the high speed command generated by the computer to equal the high speed command generated by the manual speed control.

Again, since each computer is different, it is important to re-run the XMLTweak program whenever you connect a new computer to your PCNC 1100.

9.5.7 Using a Non-standard Printer Port

There are several reasons why you may wish to use a different printer (parallel) port from the one on your PC's motherboard – for example a PCI bus plug-in card. In this case you will almost certainly need to reconfigure the PCNC Control Software to use a different address from the standard motherboard address which (in hexadecimal) is 0x378.

Use this procedure.

1. Close the Control Software if you have it open.
2. Click on the Start Menu and choose Control Panel
3. On the top left-hand side, click on Switch to Classic View. **Note:** If it says “Switch to Category View” that means you have already switched to “Classic View.”
4. Double click the System icon.
5. Choose the “Hardware” tab and click on “Device Manager.”
6. Find the “Ports (COM & LPT)” icon and click on the plus sign (figure 9.13 shows the result of these steps).
7. Double click on “ECP Printer Port (LPT1)” – wording may differ depending on your system. If you are looking for a plug-in card then the card name will probably appear in the title.
8. Choose the “Resources” tab and take note of the first number under “Setting” that corresponds with the first “Resource Type” I/O Range (figure 9.14). This shows the standard value for a motherboard port. A PCI post will not start with a zero – it will probably be something like CD80 (as it is a hexadecimal number it can have digits A, B, C, D, E or F in it). Write down the four numbers and/or letters. This is your “Printer port address in Hex.”

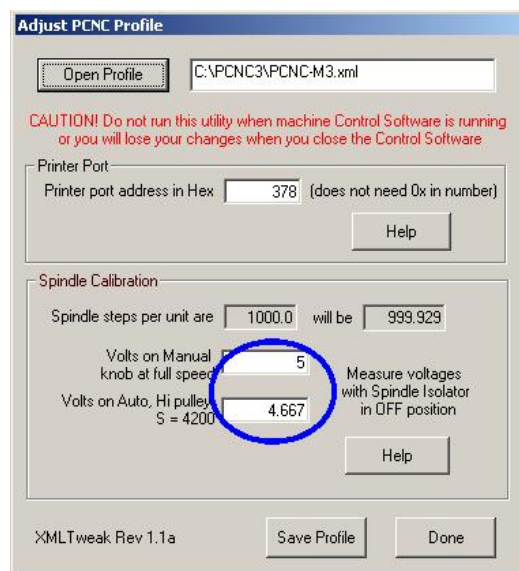


Figure 9.12 – The measured voltages

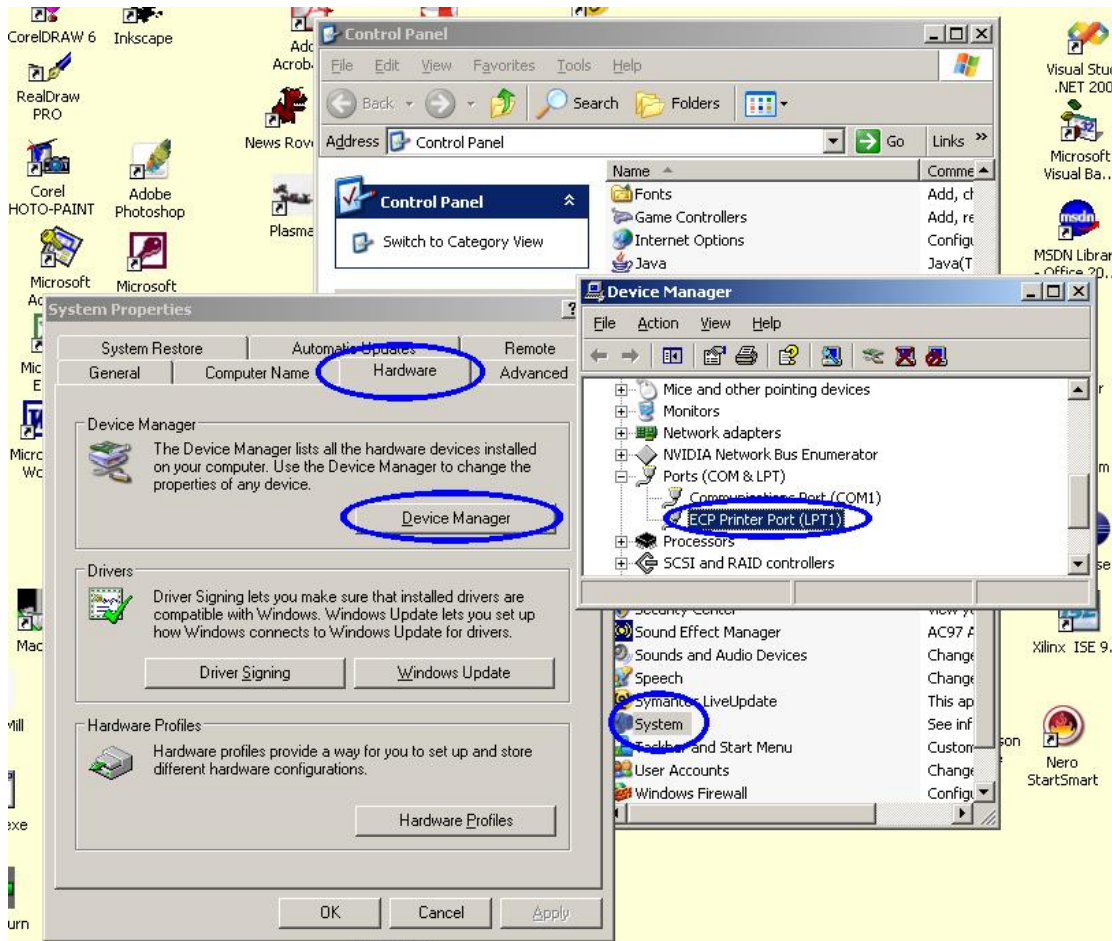


Figure 9.13 – Finding printer port

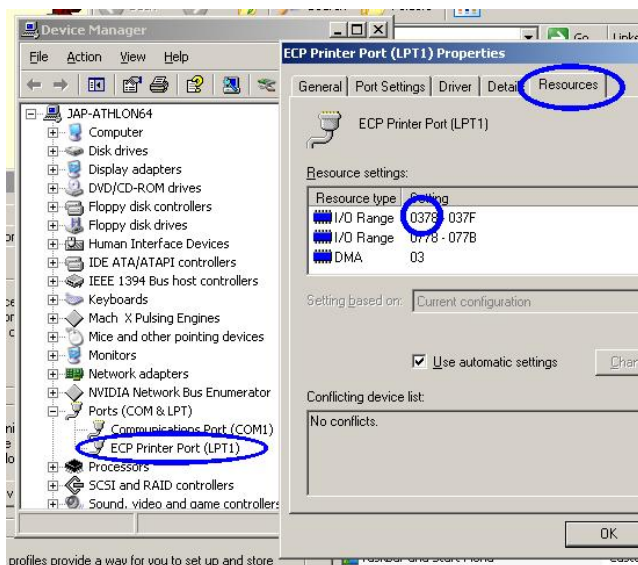


Figure 9.14 – IO address in Resources

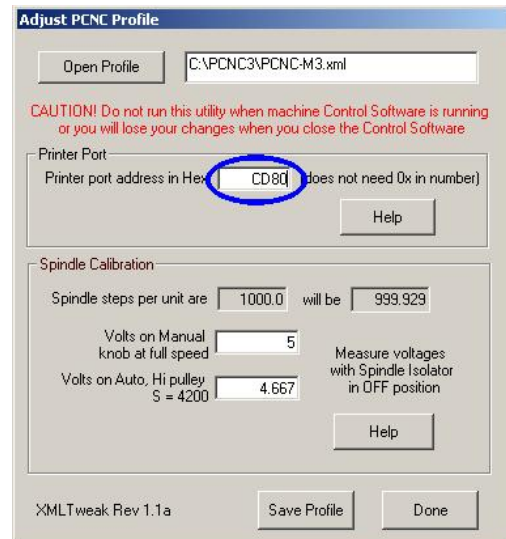


Figure 9.15 – Entering port address

9. Finally, run the XMLTweak program and open the PCNC3 profile (details in section 9.5.6 above) and enter this number under the “Printer Port” heading (figure 9.15).

9.5.8 Troubleshooting

9.5.8.1 Mechanical

The mechanical design and assembly is conventional in all aspects. Adjustments should be simple to make, but rarely needed. The exploded views are your primary resource for understanding assembly and adjustment. We encourage you to contact Tormach if additional information is needed. Specific items to be aware of include the following:

- The Z-axis is held up by the Z-axis ballscrew and motor. Decoupling the motor will result in a dangerous and destructive fall, with the entire spindle head crashing into the table.
- X-, Y- and Z-axis ballscrews are mounted on angular contact bearings. The preload adjustment on these bearings affects the axis backlash and bearing wear. Spanner wrenches are very helpful to adjusting the locknuts and setting the tension. Tormach offers the correct spanner wrenches (PN 30485) if you need to adjust or replace these bearings
- X-, Y- and Z-axis ballscrew nuts are preloaded using precision ground spacers. These are factory set and cannot be adjusted.
- X-, Y- and Z-axis slide ways each have a tapered gib plate, locked in place at each end. Tapered gibs can act as wedges and jam the machine if the lock screws are not tight.
- When correctly adjusted for preload, sustained high spindle speed will bring the spindle bearings to about 200°F (93°C). This is a normal condition. Higher preload in the spindle bearings will result in even higher temperatures and excessive wear.
- Major castings are fitted or scraped for precision fits and machine accuracy. Exchanging parts between machines or replacing parts can affect the machine accuracy. Dowel pins are used for alignment between parts. The dowel holes are drilled after alignment between parts and are unique to the original parts. Replacement parts cannot use the original dowel holes.



The following tightening torques are appropriate to the metric fasteners used in the PCNC:

Diameter (mm)	Torque (Nm)
5	7
6	12
8	30
10	55
12	100
14	160
16	245
20	480

9.5.8.2 Electrical

Electrical Service: Certain service and troubleshooting operations require access to the electrical cabinet while the electrical power is on. Only qualified electrical technicians should have perform such operations.

Many electrical problems are self-apparent. Tracing electrical problems can be done with a combination of the machine control software, the LED indicators within the machine cabinet and the machine actions.

The operating software has colored rectangular indicators, referred as LEDs, on various screens to indicate output or functional status. The Diagnostics screen also has indicators for X, Y and Z home/limit switches and accessory input status. These are useful to determine if the input is operational.

PCNC 1100 Fuses			
ID	Amp	Function	Location
FU1	15 A	Main Power	Rail Mounted in Cabinet
FU2	15 A	Main Power	Rail Mounted in Cabinet
FU3	0.6 A	Control Power	Rail Mounted in Cabinet
FU4	6.3 A	Coolant & Computer	Control Board PN 30162
FU5	1A	Control Board Power	Control Board PN 30162
F1X	6.3A	X-axis Power	DC Bus Board
F2Y	6.3A	Y-axis Power	DC Bus Board
F3Z	6.3A	Z-axis Power	DC Bus Board
F4A	6.3A	A-axis Power	DC Bus Board
	20A	Spindle Drive	VFD PN 30163

There are also various physical LED indicators within the electrical cabinet. Among these are:

- VFD Red Enable – Indicates the spindle drive is ready to turn;
- VFD Red Fault – Indicates a fault condition on the spindle drive;
- VFD Green – Indicates power to the spindle drive;
- VFD Yellow – Indicates the spindle drive is torque (current) limited;
- DC Power LED – Indicates voltage on the DC bus, power to axis drivers;
- X-, Y-, Z- and A-axis drivers – Indicate power to each individual drive;
- Control Board LED1 – Indicates power to the control board;
- Control Board D1 – Blinking indicates speed signal for the spindle.

Beyond the obvious problems of broken limit switches, blown fuses and damaged motor cables, we suggest contacting Tormach directly for advice on diagnosing or correcting electrical problems.

9.6 Preparation for Transport

It will generally be straightforward to transfer the machine to a pallet and transport it in one piece.

The Z-axis should be lowered and the spindle supported from the table on a box or solid block of timber. The Z-axis should be under a relatively neutral force, not under stress by being parked with a high downward force, nor held up by the ballscrew. Run it down to a wooden block, but do not crunch it hard into the block.

If required, the stepper motors for the axes can be removed following the reverse of the procedure described for installing the Y-axis drive. You can also remove the Z-axis motor to reduce the overall height of the machine. Take care to secure the motors after they are removed so strain is not placed on the wiring.

All bare metal surfaces should be oiled before moving the machine as a means of protection against condensation and corrosion.

9.7 Disassembly for Transport

Warranty is void if the machine is disassembled. Tormach recognizes that there are situations where users need to disassemble their mill and has made provisions in the design of the mill to facilitate this. Never the less, Tormach cannot responsible for alignment, precision and operating functions if the machine is disassembled. Test your machine before disassembling it.

The major sub-assemblies of spindle head, column and base are bolted and dowelled together so the machine can be separated into smaller components to meet very challenging transport problems. Note, however, that this entails disconnecting wiring and the lubrication lines. We recommend taking a very large number of photographs from all angles including detail photographs of any wires or oil lines that will be take apart. Dowel pins must be removed before the bolts on disassembly. Dowel pins must be installed before the bolts on assembly.

Tormach strongly recommends that all precision sliding and rotating joints remain intact during disassembly. This means that you should not remove ballscrews, bearings or separate sliding joints. For example, in reference to the head and column exploded view (drawing D40134), you should not separate item 82 (Z-axis slide) from item 75 (column). Instead you should separate item 19 (Head Casting) from item 82 (Z-axis slide).

Detailed advice should be sought from Tormach support.

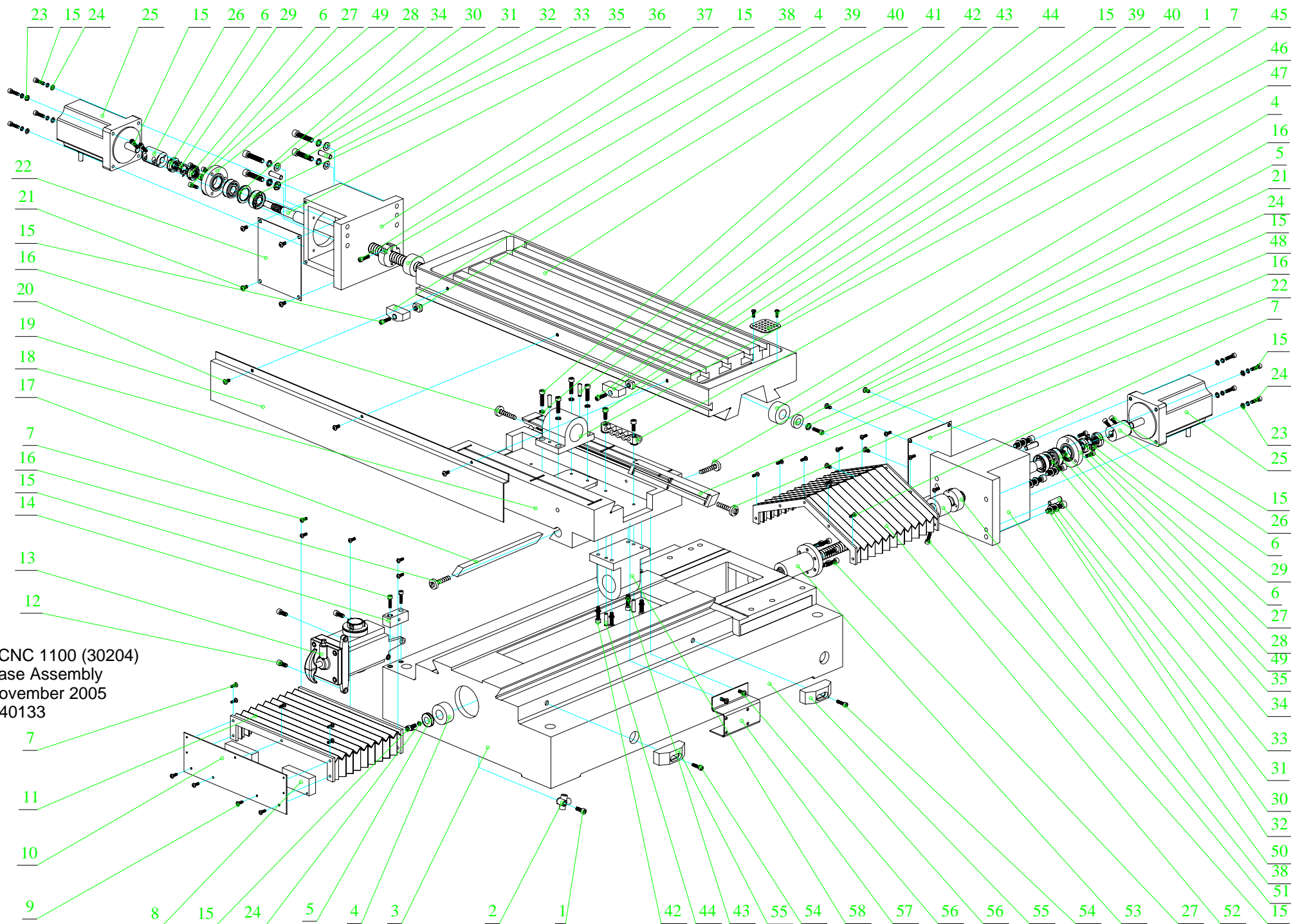
10. Appendices

10.1 Appendix 1 – Not Used

10.2 Appendix 2 – Exploded Parts Views

The following pages identify the mechanical and electrical components of the PCNC 1100, cross referenced to part numbers.

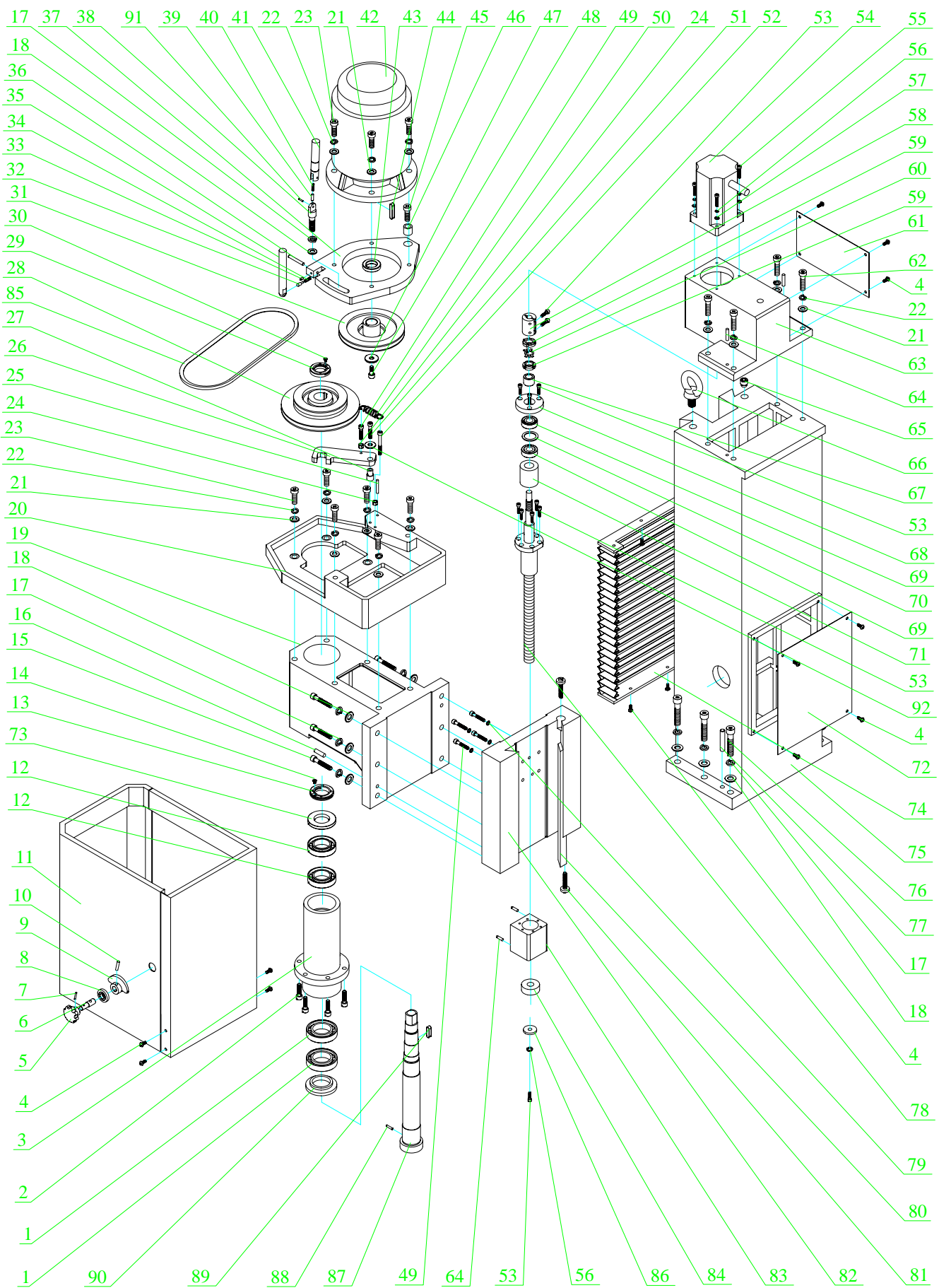
PCNC 1100 (30204)
Base Assembly
November 2005
D40133



Parts List for PCNC 1100 – Drawing D40133 (Lower Exploded) – November 2005					
ID	Number	Description	ID	Number	Description
1	30397	Screw M6x25	30	30425	Washer 8mm
2	30398	Manifold	31	30426	Lock washer 8mm
3	30399	Base	32	30427	Screw M8x40
4	30400	X & Y axis rubber bumper	33	30428	Pin 8x30
5	30401	Washer	34	30429	X & Y axis spacer
6	30402	Nut M14x1.5	35	30430	X, Y axis bearing ⁶
7	30403	Screw M4x12	36	30431	X axis ball screw & nut
8	30404	Pad block	37	30432	X axis motor base
9	30405	Screw M6x40	38	30433	Shaft clamp
10	30406	Y axis cover mounting plate	39	30434	Block
11	30407	Y axis front way cover ⁷	40	30435	Nut
11	30578	Y axis front way cover (ref note 7)	41	30436	Table
12	30408	Screw M6x16	42	30437	Screw 6x26
13	30409	Oil pump LK-8TL	43	30438	Washer 6
14	30410	Block	44	30439	Pin 6X25
15	30411	Screw M5x20	45	30440	Filter screen
16	30412	Screw	46	30441	X nut carrier
17	30413	Y axis gib	47	30442	Manifold
18	30414	Table slide	48	30443	X axis gib
19	30415	Mounting plate for X limit switch	49	30444	Sleeve
20	30416	Screw M5x8	50	30445	Y axis motor base
21	30417	Screw M5x10	51	30446	Bumper
22	30418	Plate	52	30447	Y axis rear way cover (ref note 7)
23	30419	Washer 5	52	30578	Y axis rear way cover (ref note 7)
24	30420	Washer 5	53	30448	Y axis ball screw & nut
25	30199	X, Y, and A axis motor	54	30449	Screw M5x25
26	30421	X & Y axis motor coupling	55	30450	Block
27	30422	Screw M5x16	56	30451	Screw M5x8
28	30423	X & Y ball screw cover plate	57	30452	Cover limit switch
29	30424	Washer 14	58	30453	Y axis carrier

⁶ Annular Bearing Engineering Council identification: ABEC7202AC/P5

⁷ The Y axis covers were changed after machine serial number 149, but dimensions should be checked as many machine owners installed a conversion kit. Earlier covers were 280mm (11") wide, later covers were 320mm (12.6") wide. Later covers were also better enforced and a more durable material. Early front and rear covers were 30407 and 30447. Later covers were 30578 for both front and rear. Conversion kit is PN 30580.



PCNC 1100 (30204)
 Column & Head Assembly
 November 2005
 D40134

Parts List for PCNC 1100 – Drawing D40134 (Upper Exploded) – November 2005					
ID	Number	Description	ID	Number	Description
1	30303	Lower spindle bearing ¹	47	30351	Pulley washer
2	30304	Screw M8x30	48	30352	Screw M8x20
3	30305	Spindle cartridge	49	30353	Screw M6x30
3A	30306	Spindle cartridge assembly ²	50	30354	Spring
4	30307	Screw M5x12	51	30355	Screw M6x40
5	30308	Knob	52	30356	Screw M6x60
5A	30309	Motor cover lock assembly ³	53	30357	Screw M5x20
6	30310	Shaft	54	30200	Z axis step motor
7	30311	Pin 3X20	55	30359	Screw M5x26
8	30312	Sleeve	56	30360	Washer 5
9	30313	Cam	57	30361	Washer
10	30314	Pin 5x26	58	30362	Clamping shaft collar
11	30315	Motor cover	59	30363	Nut M14X1.5
12	30316	Upper spindle bearing ⁴	60	30364	Lock washer 14mm
13	30317	Nut M33x1.5	61	30365	Cover plate
14	30318	Screw M5x8	62	30366	Screw M10x40
15	30319	Screw 12x50	63	30367	Z axis motor base
16	30320	Pin 8x35	64	30368	Pin 6x30
17	30321	Washer 12mm	65	30369	Screw M16x12
18	30322	Washer 12mm	66	30370	Screw M16
19	30323	Head casting	67	30371	Spacer
20	30324	Spindle motor base	68	30372	Z ball screw cover plate
21	30325	Washer 10mm	69	30373	Z axis ball screw bearing ⁵
22	30326	Washer 10mm	70	30374	Spacer
23	30327	Screw M10x30	71	30375	Z axis ball screw upper bumper
24	30328	Nut M6	72	30376	Column cover plate
25	30329	Pin 5X35	73	30377	Spacer
26	30330	Spindle lock	74	30378	Z axis way cover
27	30331	Spindle lock pivot	75	30379	Column
28	30332	Spindle pulley	76	30380	Screw M12x60
29	30333	Nut M27x1.5	77	30381	Pin 10X55
30	30334	Screw M4x8	78	30382	Z axis ball screw & nut
31	30335	Motor pulley	79	30383	Washer 6
32	30336	Handle pin	80	30384	Z axis gib
33	30337	Set screw M5X8	81	30385	Screw
34	30338	Spring 1x4.8x20	82	30386	Z axis slide
35	30339	Adjustable base handle	83	30387	Z nut carrier
36	30340	Adjustable base pivot pin 6X45	84	30388	Z axis screw lower bumper
37	30341	Adjustable base spindle motor	85	30389	Belt 3V280 Gates
38	30342	Adjustable base clamp bolt	86	30390	Unused
39	30343	Pin	87	30391	Spindle R8
40	30344	Spring 1x4.8x20	88	30392	Pin
41	30345	Adjustable base clamp handle	89	30393	Key 8X26
42	30346	Spindle motor	90	30394	Lower spindle spacer
43	30347	Spacer	91	30395	Cylindrical pin 4X16
44	30348	Key 8X40	92	30396	Big washer 6
45	30349	Screw M12x45		30507	Drawbar for R8 taper
46	30350	Sleeve		30506	Drawbar for BT30 taper

¹ Annular Bearing Engineering Council identification: DT7008/DT (double tandem pair)

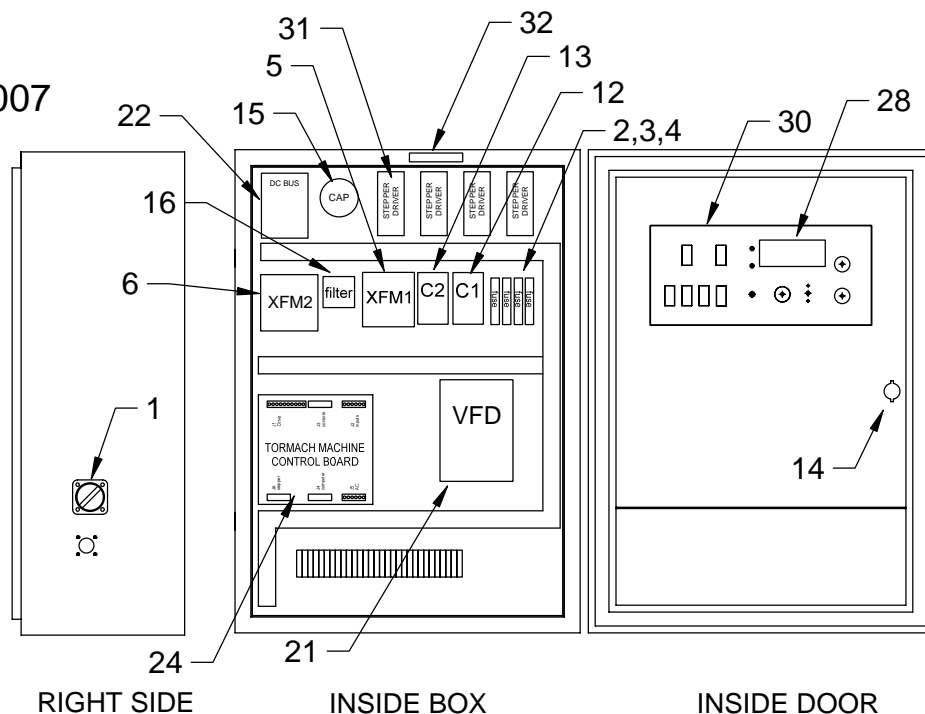
² Spindle Cartridge Assembly for R8 includes callout numbers 1,3, 12,13,14,73, 87, 88, and 90.
For BT30 Spindle Cartridge Assembly, use PN 30505

³ Motor Lock Assembly includes callout numbers 5,6,7,8,9, and 10

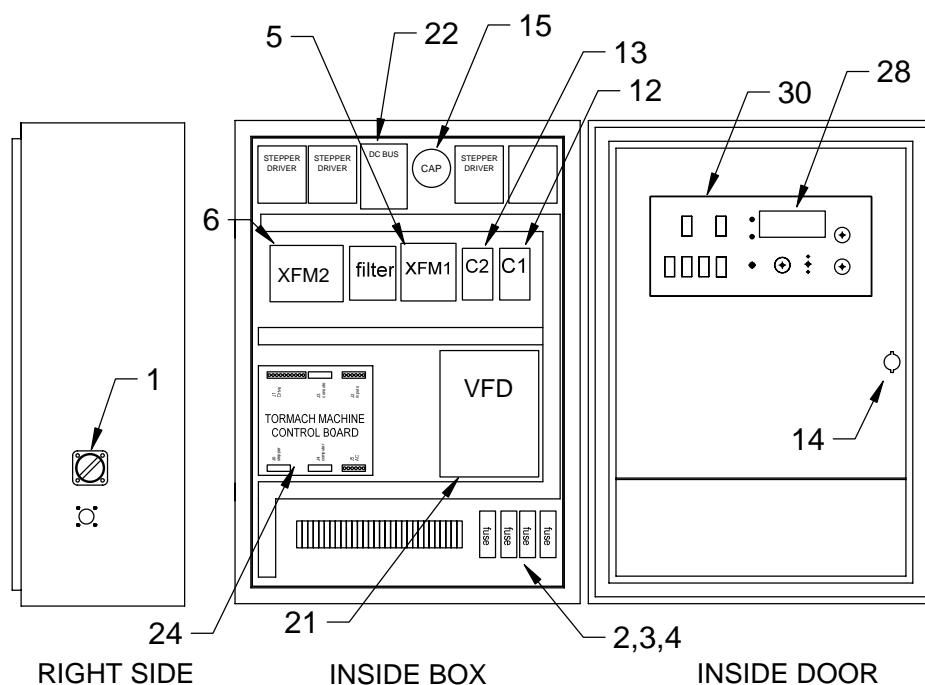
⁴ Annular Bearing Engineering Council identification: D7007C/DT (double tandem pair)

⁵ Annular Bearing Engineering Council identification: ABEC7202B/P5

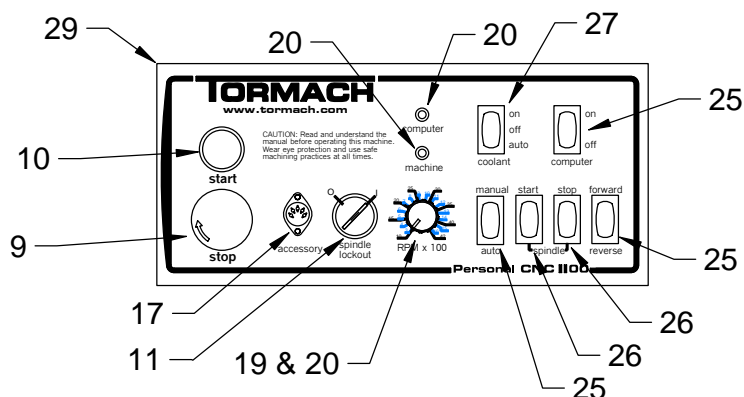
CABINET
(4 digit serial numbers)



CABINET
(3 digit serial numbers)



OPERATOR PANEL (all serial numbers)



Parts List for PCNC 1100 – Drawing D40135 (Electrical Overview) – December 2007					
ID	Number	Description	ID	Number	Description
1	30454	Disconnect switch	25	30171	On-Off Rocker
2	30455	Fuse FU1, FU2 ⁸	26	30172	(On)-Off Momentary Rocker
2	30510	Double Fuse Block	27	30173	On–Off-On Rocker
3	30456	Fuse FU6 ⁹	28	30160	Connection Board
3	30511	Single Fuse Block	29	30208	Operator Console Overlay
4	30724	Fuse FU3 ¹⁰	30	30209	Operator Console Backplate
4	30511	Single Fuse Block	31	30541	Stepper Driver
5	30458	Transformer XFM1	32	30625	Fan, 115 VAC
6	30459	Transformer XFM2			
9	30462	Estop PB1			
10	30463	Push button PB2			
11	30464	Key switch with keys SW7		30627	Flex Conduit 16 mm OD
12	30465	Relay contactor C1		30628	Connector for 16 mm flex
13	30466	Relay Contactor C2		30722	Flex Conduit 12 mm OD
14	30467	Electrical cabinet latch & key		30723	Connector for 12 mm flex
15	30468	Capacitor		30728	Flex Conduit 10 mm OD
16	30469	Filter		30729	Connector for 10 mm flex
17	30178	Accessory / Probe Port		30470	Tormach Logo -vinyl
18	30179	Potentiometer		30471	PCNC 1100 Logo -vinyl
19	30181	Knob		30222	Label, Belt Position
20	30180	LED		30223	Label, Machine Safety
21	30163	VFD Motor Driver		30224	Label, Dual Power Safety
22	30661	DC Bus Board		30225	Label, Retain Voltage Safety
23	30164	Stepper Driver		30684	J4 cable (computer)
24	30162	Machine Control Board		30685	J3 cable (console)
25	30171	On-Off Rocker		30686	J6 cable (axis drivers)

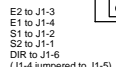
Limit and Guard Switches – August 2007	
Position	Description
X limit	PN 30742 – all machines
Y limit	PN 30535 – machine number 75 and earlier. Later machines use PN 30461 for switch only, PN 30536 for switch and welded enclosure/mount.
Z limit	PN 30461 – all machines
Spindle Door	PN 30461 on machine number 220 and earlier. PN 30577 on all later machines.
Part Number	Description
30461	Roller switch identified as 2" x 1.75" x 0.75" plastic and metal
30535	Roller switch identified as painted metal body 2" x 2" x 3.75"
30577	Push switch identified as plated metal body 2" x 2.75" x 0.75"
30742	Roller switch

⁸ Metric size is 10x38 mm, 15A. Use Bussmann KTK-15, Littlefuse KLK-15, or Ferraz ATM-15

⁹ Metric size is 10x38 mm, 6A. Use Bussmann KTK-6, Littlefuse KLK-6, or Ferraz ATM-6

¹⁰ Metric size is 10x38 mm, 0.6A. Use Bussmann KTK-6/10, Littlefuse KLK-.6, or Ferraz ATM-0.6

(3 digit serial numbers)



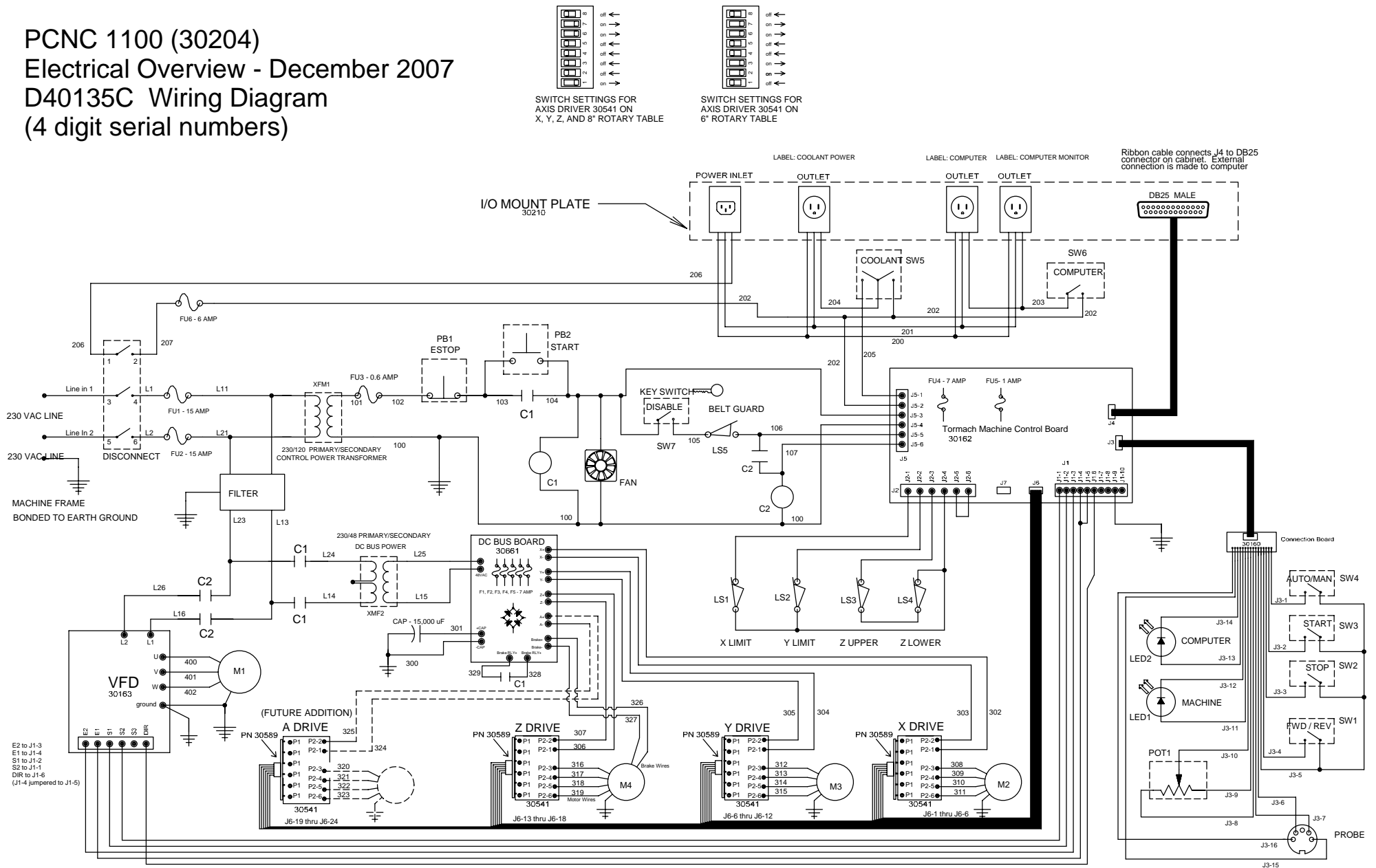
Note: More detail can be seen by viewing this document on a computer screen as a PDF file.

PCNC 1100 (30204)

Electrical Overview - December 2007

D40135C Wiring Diagram

(4 digit serial numbers)



Note: More detail can be seen by viewing this document on a computer screen as a PDF file.

[illegible]

10.3 Appendix 3 – Optimization of Windows XP

The PC running your mill does not simply send G- & M-codes down to the mill; rather it is intimately involved with control of the axis motors and spindle. It reviews and updates electrical signals to every axis motor 25,000 times each second. If your PC is busy thinking about other things, there can be problems in motion control.

We advise you not to cut a part with the control computer connected to a network.

The steps listed below will optimize Windows XP. There are a large number of esoteric Windows functions which, if enabled, can cause problems. You will know if you have difficulties if machine motion sounds rough or if a long move makes a periodic ticking sound.

In difficult cases please consult Tormach for more detail changes that could be made.

10.3.1 Remove Unnecessary Services and Startup Programs

Background: There are a variety of programs that can run in the background, stealing CPU power. These programs may watch your keystrokes, track communications or try to seek out networks. New computers are often laden with “freebee” versions of programs that are loaded when the computer starts. Things to remove include QuickTime, RealPlayer, AOL messenger, QuickBooks, Instant Messenger, camera, video software, music software and just about any application software that you do not use or recognize. Your mill computer should be lean and clean and void of any unnecessary programs.

1. Click *Start* button.
2. Click *Run*.
3. Type *MSCONFIG* and press *Enter*.
4. Click *Services* tab.
5. On the lower line, check the box for *Hide All Microsoft Services* (step 1), then *Disable All* (step 2), finally click on *Apply* (step 3)
6. Now click on the *Startup* tab
7. Click on *Disable All* then *Apply*. Some people prefer to have the PCNC software called out in the *Startup* group so the control program is running as soon as the computer is turned on. In this case leave that as the only *Startup* program.

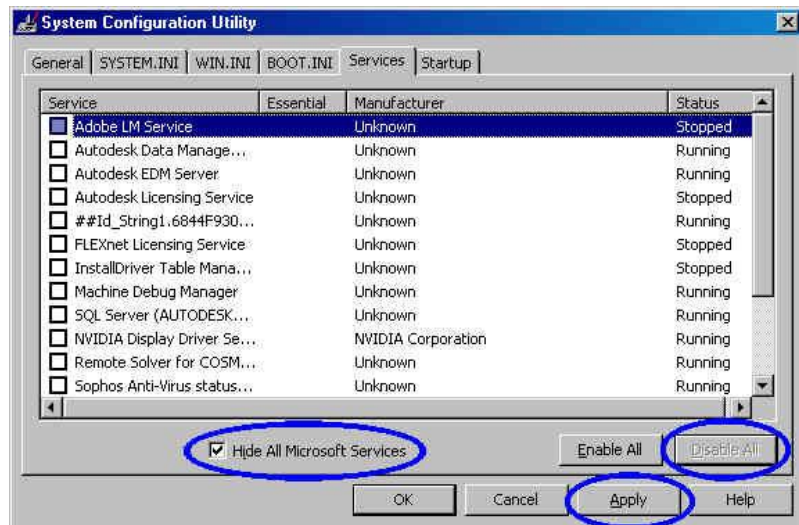


Figure 10.1 – Disable Services

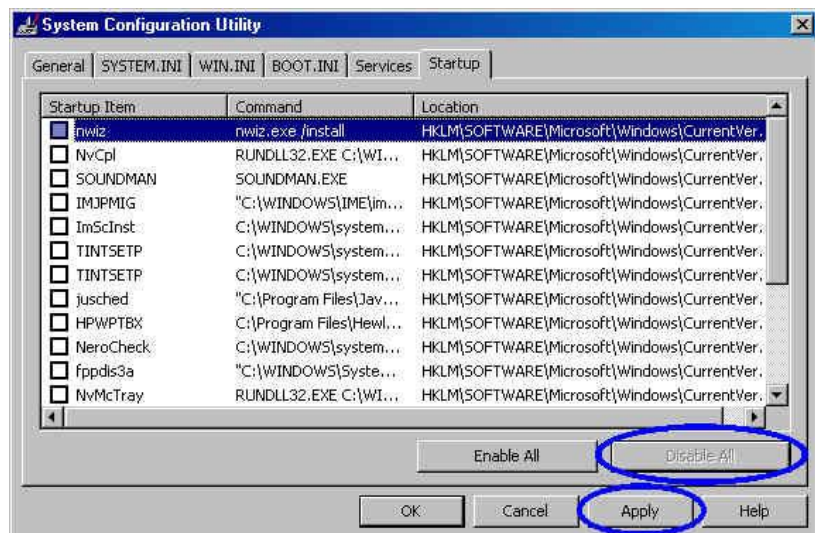


Figure 10.2 – Disable Startup programs

8. Click OK to complete.

10.3.2 **Disable Power Management**

1. Right-click on your desktop and then click *Properties*.
2. Click on the *Screen Saver* tab.
3. Set Screensaver to None.
4. Press the *Power* button near the bottom.
5. Set all options to NEVER shut down automatically!

10.3.3 **Disable Automatic Updates**

Background: Automatic updates can initiate a CPU stealing background task. This can create havoc if it occurs while the machine is running a program. Some updates force a system re-boot after a timeout period. This is disastrous when running a job with the screen unattended.

1. Right Click *My Computer* and select *Properties*.
2. Click *Automatic Updates* tab.
3. Uncheck *Keep my computer updated*.
4. Click *OK*.

10.3.4 **Set Computer to Standard PC not ACPI PC**

Background: This optimization procedure is not needed on most computers, but it is required on some. We recommend this procedure only be applied last and only if necessary. Advanced Configuration and Power Interface) is a power management specification that allows the operating system to control the how power is applied to the computer's devices. For example, an ACPI PC can be turned off completely by using Windows Shutdown function. With a Standard PC (not ACPI), the Windows Shutdown function will close programs and prepare the operating system, then open a window with the statement "You can now turn off your computer". A Standard PC can only be turned off at the power switch on the box itself, not through the operating system.

1. Right Click *My Computer* and select *Properties*.
2. Click *Hardware* tab.
3. Click *Device Manager* button in the middle.
4. Double click *Computer*.
5. Right click on *Standard ACPI PC* and choose *Update Driver*.
6. Choose *Install the software from a Specific Location (Advanced)*.
7. Click *Next*.
8. Choose *Do not search. I will choose driver to install*.
9. Click *Next*.
10. Choose *Standard PC* from the listing.
11. Click *Next*.
12. Click *OK*.

When you re-boot Windows will have to find and load many drivers. The Standard PC drivers for the video, hard disk, keyboard and other devices are all different than the ACPI drivers. This can take a long time and it may appear to have frozen for many minutes during the process. It may require re-booting several times.

10.4 Revision history

Rev C3.1	6 January 2008	Clarification of tapping operations. Simplification of computer optimisation process. Addition of new electrical component part numbers.
Rev C2-3	19 August 2007	Clarification of cutting FirstPart. Additional details of limit switches in exploded views and power to coolant pump and computer
Rev C2-2	7 August 2007	Stepper Driver switch setting details on schematic diagram. The probe and tool setter screens added. Addition of information on Tapping and support macros. Addition of G68/G69. Tool offsets revised to describe "Master Tool as longest tool" procedure.
Rev C1-4	4 May 2007	Update to some illustrations and the consistency of paragraph labeling
Rev C1-3	2 May 2007	Stylistic standards tightened, detailed drafting improved
Rev C1-2	1 May 2007	More consistent naming of accessories used
Rev C1-1	18 April 2007	First draft of Mach3 version

11. Index

Hint: Where there is a choice, most index entries are made using the name of a thing (e.g., tool offset) rather than an action (e.g., measuring) so you will get better results thinking about the part of the machine on which you want information. Thus looking for “Tool offset – measuring” will give better results than looking for “Measuring – tool offset.” For important information both entries will probably appear.

If you have difficulty because you tried to look something up and the index entry was missing, please take a moment to e-mail tech@tormach.com with a note of (a) the words you were looking up and (b) where in the manual you found the information you wanted – assuming you did!

A

Absolute distance mode	
G90	7-26
Absolute IJ mode	7-14
Absolute machine coordinates	
G53 - move in	7-20
Accessory socket	2-13
on control panel	5-2
Address	
setting non-setting non-standard printer port	9-10
Adjustable parallel gauge	6-4
Arc - center format	7-14
Arc - radius format	7-13
Arc at feedrate	
G02/G03 defined	7-13
Arc motion	
defined	7-2
Assembly Y-axis drive	2-4
Axis controls family	
described	5-6
Axis coordinate DRO	
described	5-6
Axis jogging	
control family	
described	5-7
keyboard	
Continuous	5-7
Step	5-7
MPG	5-7
rate override	5-7
Step selected by Ctrl key	5-8
with Tormach Jog/Shuttle Controller	5-8

B

Base assembly - exploded view	5
Bearings, ballscrew	
adjustment of preload	9-5
Binary operations	
defined	7-7
Block	
format of code	7-5

Block Delete	
action of	7-4
Block delete switch	5-11
Blue Screen of Death	
action to avoid	2-11
Boring and reaming canned cycle	
G85	7-25
Boring manual retract canned cycle	
G88	7-25
Boring with dwell and retract canned cycle	
G89	7-25
Boring with dwell canned cycle	
G86	7-25

C

Cabinet/stand	2-5
CAD	
2d, 2 ½D and 3D explained	4-2
CAD/CAM software	
what to look for	4-2
Calculator	
feeds and speeds	5-15
Cancel modal motion	
G80 explained	7-22
Canned cycle return level	
G98/G99	7-27
Canned cycles	7-22
in-between motion	7-23
preliminary motion	7-23
repeats by L word	7-22
retract defined by R word	7-23
sticky numbers	7-22
Center format arc	7-14
Centering microscope methods	6-6
Circular pocket	
G12/G13	7-16
Code definition syntax explained	7-11
Column and head assembly	3
Comments	
defined	7-8
Compensation - tool diameter	
alignment move	6-15
Computer LED	

Appendices

on control panel.....	5-2
Computer mounting	
accessories	2-3
arrangements	2-3
Computer ON/OFF	
control panel switch.....	2-12, 5-1
Computer requirements.....	2-7
Concept behind PCNC.....	1-3
Connecting control computer to PCNC.....	2-11
Constant velocity mode	
G64 - setting.....	7-21
purpose of explained.....	7-3
Control panel controls explained	2-12
Control Software	
Components of	2-9
screens	5-5
Controlled point	
defined	7-1
Conversationl programming.....	4-9
Coolant	
control of.....	7-3
control panel switch.....	2-13
importance of in CNC.....	2-2
M07 - mist on.....	7-29
M08 - flood on	7-29
M09 - all off.....	7-29
Coolant controls	
on control panel.....	5-2
Coordinate rotation by probing	
setting coordinate rotation.....	8-10
Coordinate system rotation	
G68 – setting and G68 - clearing.....	7-21
Coordinate systems	
and offsets	6-1
reference definitions	7-5
Co-ordinated linear motion	
defined	7-2
Copyright statement.....	1-4
Current position	
defined	7-3
Cutter compensation	
material edge contour	6-13
tool diameter	
entry move	
general.....	6-15
first move.....	6-15
entry move	6-16
tool path contour.....	6-14
Cutter diameter compensation	
through CAD/CAM.....	6-12
through control software.....	6-12
Cutter radius compensation	
G40/G41/G42 defined	7-19
Cycle Start button	5-10

D

Demonstration program	
running.....	3-2
Diameter compensation LED	
described	5-7
Digitizer probe.....	8-3

Digitizer probe interface	8-12
Direction of jogging.....	2-16
Disclaimer of liability	1-4
Display mode button.....	5-12
Diver test program.....	2-10
Drilling canned cycle	
G81	7-23
Drilling with dwell canned cycle	
G82.....	7-24
Dwell	7-3
G04 - defined	7-15

E

Electrical overview diagram.....	7
Electrical specifications	9-2
EStop button.....	2-13
on control panel.....	5-2
Exact stop mode	
G61 - setting.....	7-21
purpose of explained.....	7-3
Excecution of words	
order of	7-31
Exploded view	
base assembly.....	5
column and head assembly.....	3
Expressions	
defined	7-7

F

F word -feedrate	7-31
Feed & Speed calculator	5-15
Feed and speed override	
controlled by M48/M49	7-29
Feedrate	
defined	7-2
display DROs	
described.....	5-9
F word to set.....	7-31
inverse time - G93	7-27
units per minute - G94	7-27
units per rev.....	5-9
units per rev - G95.....	7-27
Feedrate control family	
described	5-9
Feedrate override DRO	
described	5-9
Feedrate units per rev - G95	7-27
File control family	5-12
First part	
making	3-1
Fixing to stand.....	2-6
Fixture coordinate select	
G54-G59 defined.....	7-21
Fixture coordinate systems - setting - G10.....	7-15
Fourth axis	
diameter compensation of feedrate.....	8-1
engraving on a cylinder.....	8-2
gear cutting	8-2
referencing and zeroing	8-1
Fourth axis on PCNC.....	8-1
From idea to a part.....	4-1

G

G & M codes	
overview	4-1
G00 - rapid linear motion	7-11
G01 - linear feedrate move	7-13
G02 - clockwise arc	7-13
G03 - counterclockwise arc	7-13
G04 - dwell	7-15
G10 - set coordinate systems	7-15
G12 - circular pocket	7-16
G13 - circular pocket	7-16
G15 - exit Polar mode	7-16
G16 - enter Polar mode	7-16
G17 - select XY plane	7-16
G18 - select XZ plane	7-16
G19 - select YZ plane	7-16
G20 - inch units - setting	7-17
G21 - millimetre units - setting	7-17
G28 - return to home	7-17
G28.1 - reference axes	7-17
G30 - return to home	7-17
G31 - and probe hardware options	8-4
G31 - straight probe	7-17
G40 - cutter radius compensation - Off	7-19
G41 - cutter radius compensation - Left	7-19
G42 - cutter radius compensation - Right	7-19
G43 - enable tool length offset	7-19
G44 - enable tool length offset	7-19
G49 - disable tool length offset	7-19
G50 - clear axis scale factors	7-20
G51 - set axis scale factors	7-20
G52 offsets	7-20
G53 - move in absolute machine coordinates	7-20
G54 - select fixture 1	7-21
G55 - select fixture 2	7-21
G56 - select fixture 3	7-21
G57 - select fixture 4	7-21
G58 - select fixture 5	7-21
G59 - select any fixture	7-21
G61 - set exact stop mode	7-21
G64 - set constant velocity mode	7-21
G68	
define coordinate system rotation	7-21
G69	
Clear coordinate system rotation	7-21
G73	
- high speed peck drilling canned cycle	7-21
pullback DRO	7-22
G80 - cancel modal motion	7-22
G81 - drilling canned cycle	7-23
G82 - drilling with dwell canned cycle	7-24
G83 - peck drilling canned cycle	7-24
G85 - boring and reaming canned cycle	7-25
G86 - boring with dwell canned cycle	7-25
G88 - boring manual retract canned cycle	7-25
G89 - boring with dwell and retract cycle	7-25
G90 - absolute distance mode	7-26
G91 - incremental distance mode	7-26
G92 - workpiece offsets	
interaction with parameters	7-26
G92 offsets	7-26

G93 - feedrate inverse time	7-27
G94 - feedrate units per minute	7-27
G98 - canned cycle return level	
to old Z	7-27
G99 - canned cycle return level	
to R word	7-27
G-code	
running	4-6
G-code display control	5-11
Goto toolchange	5-11

H

Home - return to G28/G30	7-17
--------------------------------	------

I

I/J Modes setting	5-16
IJ mode - "Absolute"	7-14
IJ mode - Increments	7-14
Inch units	
G20 - setting	7-17
Incremental distance mode	
G91	7-26
Incremental IJ mode	7-14
Inhibits and overrides control family	5-15
Installation	
errors after	2-10
of control software	2-9
of driver	
manual	2-11
Intellectual property statement	1-4
Intelligent labels	
described	5-6
Intended use	
policy if use is out of scope	9-1
statement of	9-1
Isolator	<i>See</i> - Main switch

J

Jerky motion with short lines	
Constant velocity mode to avoid	7-3
Jog/Shuttle controller	5-8
Jogging	
direction of	2-16
Jogging axes	2-15
Jogging pendant	5-16

L

Laser finder methods	6-6
Learning and Training	2-3
License statement	1-4
Lifting onto stand	2-5
Lighting for machine	2-2
Line	
format of code	7-5
Line DRO	
described	5-10
Line number	
format of	7-5
Linear axes	
defined	7-1
Linear feedrate move	

Appendices

G01 defined.....	7-13
Location of machine	2-1
Location to give safety.....	2-2
Locking	
spindle.....	5-3
Loop control family	5-13

M

M00 - program stop	7-27
M01 - optional program stop.....	7-27
M02 - program end	7-28
M03 - spindle clockwise	7-28
M04 - spindle counterclockwise.....	7-28
M05 - stop spindle	7-28
M07 - mist coolant on	7-29
M08 - flood coolant on.....	7-29
M09 - all coolant off	7-29
M30 - program end	7-28
M48 - feed and speed override on.....	7-29
M49 - feed and speed override off.....	7-29
M871	
tapping canned cycles	7-30
M87x	
tapping canned cycles	7-30
M98 - subroutine call.....	7-29
M99 - subroutine return	7-30
Machine coordinates	
displaying on axis DROs	5-7
G53 - move in	7-20
Machine modes	
defined	7-9
Machine ON LED.....	2-13
Macro M-codes.....	7-30
Macros	
overview on writing.....	7-30
Main switch.....	2-11, 5-1
Maintenance	
Bearing preload adjustment.....	9-5
lubrication	9-3
protection from rust	9-3
speed calibration.....	9-8
way covers	9-4
Z axis adjustment.....	9-4
Manual Data Input to control Software.....	2-15
Master tool	
choosing.....	6-9
for tool offsets	6-8
M-code	
macros.....	7-30
M-codes - built in	
summary table	7-27
MDI	2-15
Mechanical specifications	9-2
Messages	
from part program, defined	7-9
Millimetre units	
G21 - setting.....	7-17
Mirroring parts	7-1
Modal groups	
defined	7-10
Modal motion, cancelling	

G80 explained	7-22
Mode	
of jogging	2-15
Modes	
machine - defined	7-9
MSG,	
string introduces an operator message	7-9

N

Number	
format of	7-7

O

Offsets	
G52	7-20
G92	7-26
tool.....	6-7
Offsets and coordinate systems	6-1
Operators - binary	
defined	7-7
Operators - unary	
defined	7-8
Operator's panel.....	5-1
Optimising Windows	2-8
Optimising Windows for PCNC.....	12
Optional program stop	
M01	7-27
Optional Stop	
action of	7-4
Optional Stop switch.....	5-11
Order of G-code items on line	7-9
Override	
for feed and speed - disabling.....	7-4
Override feed and speed	
controlled by M48/M49	7-29

P

Parameter	
setting value of	7-8
using value of	7-7
Parameters	
predefined	7-5
Part program	
repeating indefinitely - M47.....	7-29
repeating indefinitely - M99.....	7-30
running controls family	
described.....	5-10
Part-program	
making with Wizards	4-9
Pause button	5-10
PCNC	
choosing location of.....	2-1
computer requirements	2-7
connecting to control computer	2-11
control panel.....	5-1
control panel controls explained.....	2-12
fourth axis option.....	8-1
operator's panel.....	5-1
receiving and unpacking	2-4
tool changing.....	5-3
PCNC computer mounting arrangements.....	2-3

Appendices

PCNC concept	1-3
Peck drilling canned cycle	
G83	7-24
Peck drilling canned cycle – high speed	
G73	7-21
Pendant - jogging	5-16
Performance expectations	1-3
Plane selection	
G17/G18/G19 defined	7-16
Polar mode	7-16
Power supply requirements	2-3, 2-6
Preface	1-1
Printer port	
setting non-standard address	9-10
Probe	<i>See</i> Straight probe
calibration	8-11
Probes and Tool setters	8-3
Probing	
comprehensive X/Y	8-8
finding center of bore	8-5
finding position of a face	8-5
for corner of a vise	8-6
setting coordinate rotation	8-10
simple X/Y	8-4
surface Z	8-7
Z axis	8-6
Profile	
display name of profile in use	5-6
Program	
error handling	7-31
Program end	
M02/M30	7-28
Program extrema	5-11
Program stop	
M00	7-27
Pullback DRO	
G73	7-22
Pulleys	
spindle speed control	
described	5-8
R	
R8 tooling	
inserting and removing	5-3
Radius format arc	7-13
Rapid motion	
G00 definrd	7-11
Reaming and boring canned cycle	
G85	7-25
Re-boot during installation	
how to manually uninstall driver if you fail to do it	
.....	2-11
reason for	2-9
Receiving new machine	2-4
Reference - G28.1	7-17
Referenced LED	
described	5-6
Referencing	
importance of	3-2
introduction to	3-1
Regen button	5-12

Repeating part program indefinitely - M47	7-29
Repeating part program indefinitely - M99	7-30
Reset button	
described	5-5
Return level after canned cycle	
G98/G99	7-27
Rewind button	5-10
Roller gauge	6-3
Roll-you-own gauge	6-2
Rotational axes	
defined	7-1
Rotational diameter correction	
controls family	5-14
Run from here button	5-10

S

S word - spindle speed	7-31
Safety	
checklist	1-2
Dual supply - implications of	1-1
from machine location and mounting	2-2
further reading	1-2
general	1-1
Grounding	1-1
Safety warning	1-1
professional advice	1-1
Scale factor - on axis data - G50, G51	7-20
Scale factor DRO	
described	5-7
Scaling coordinates	7-1
Scaling parts	7-1
Screen switching buttons	5-6
Screen switching controls	
described	5-5
Selected plane	
defined	7-3
Set fixture coordinate systems - G10	7-15
Set next line button	5-10
Shuttle controller	5-8
Simulate Program Run button	5-11
Single button	5-10
Space required - minimum	2-2
Special Mach3.sys driver	
installation of	2-10
need for	2-10
specialdriver.bat	2-10
Specification	
options	9-3
Specifications	
electrical	9-2
mechanical	9-2
system	9-3
Speed and feed override	
controlled by M48/M49	7-29
Spindle	
belt changing	5-5
Changing the belt position	2-14
Control panel Forward/Reverse switch	2-13
Control panel Start switch	2-13
Control panel Stop switch	2-13
locking	5-3

Appendices

Lockout switch.....	2-13
M03 - clockwise.....	7-28
M04 - counterclockwise.....	7-28
M05 - stop.....	7-28
Manual/Auto	2-13
Spindle controls	
on control panel.....	5-2
Spindle speed	
control family described	5-8
S word to set.....	7-31
Spindle speed calibration	9-8
Stand/cabinet.....	2-5
Start button.....	2-13
on control panel.....	5-2
Status LEDs	
on control panel.....	5-2
Stop button	5-10
Straight probe	
example program.....	7-18
G31 defined.....	<i>See</i> Straight probe
Subroutine call	
M98	7-29
repeating several times.....	7-29
Subroutine label	
format of	7-6
Subroutine return	
M99	7-30
Support	
from Tormach	9-1
Switch - main	2-11
Syntax - Code definition	7-11
System specifications.....	9-3
T	
T word - tool select.....	7-31
Tapping	
sequence for threading a hole.....	8-13
Tapping heads	
description of the heads	8-13
Teach control family.....	5-13
Testing	
Control Software installation	2-10
DiverTest program.....	2-10
Tool change	
supplied M6 macros.....	7-29
Tool change position	
going to.....	7-30
Tool changing.....	5-3
Tool diameter correction	6-9
Tool length	
setting by tool setter.....	8-7
Tool length offset	
by tool setter.....	6-10
G43 - enable.....	7-19
G44 - enable.....	7-19
G49 - disable	7-19
Tool offsets	6-7
Tool select	
T word	7-31
Tool setters and probes	8-3

Toolchange position control family	5-14
Toolpath	
display looks inaccurate.....	5-11
Tormach Jog/Shuttle controller	5-8
Tormach Tooling System	
TTS.....	5-3
Touching techniques.....	6-2
Trademarks.....	1-4
Transportation of machine	9-13
Troubleshooting	
electrical.....	9-13
mechanical	9-12
TTS	
with tool offsets	6-7
TTS - Tormach Tooling System	5-3
TTS tooling	
inserting and removing	5-4
Typographics conventions in this manual	

U

Unary operators	
defined	7-8
Un-installation of driver	
manual	2-11
Units	
inch, degree and millimetre	7-3
Unpacking new machine.....	2-4
Unusual modes	
when Control Software is running.....	5-14

W

Windows	
optimisation procedures	12
Wizard	
checking code output by	4-10
Wizards	
for conversational programming	4-9
Word	
format of	7-7
initial letters	7-7
Work offsets	
by Touch buttons.....	6-5
by typing into DRO	6-5
setting X and Y by probe	6-6
using more than one.....	6-12
X and Y, setting.....	6-5

X

XMLTweak	
setting non-standard address	9-10
spindle speed calibration.....	9-8

Y

Y-axis drive	
assembling on new machine.....	2-4

Z

Z axis adjustment.....	9-4
------------------------	-----